

# **MIKE SHE USER MANUAL**

## **VOLUME 1: USER GUIDE**



10 November 2006 12:49 pm



---

## **Please Note**

### **Copyright**

This document refers to proprietary computer software which is protected by copyright. All rights are reserved. Copying or other reproduction of this manual or the related programs is prohibited without prior written consent of DHI Water & Environment (DHI). For details please refer to your 'DHI Software Licence Agreement'.

### **Limited Liability**

The liability of DHI is limited as specified in Section III of your 'DHI Software Licence Agreement':

'IN NO EVENT SHALL DHI OR ITS REPRESENTATIVES (AGENTS AND SUPPLIERS) BE LIABLE FOR ANY DAMAGES WHATSOEVER INCLUDING, WITHOUT LIMITATION, SPECIAL, INDIRECT, INCIDENTAL OR CONSEQUENTIAL DAMAGES OR DAMAGES FOR LOSS OF BUSINESS PROFITS OR SAVINGS, BUSINESS INTERRUPTION, LOSS OF BUSINESS INFORMATION OR OTHER PECUNIARY LOSS ARISING OUT OF THE USE OF OR THE INABILITY TO USE THIS DHI SOFTWARE PRODUCT, EVEN IF DHI HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. THIS LIMITATION SHALL APPLY TO CLAIMS OF PERSONAL INJURY TO THE EXTENT PERMITTED BY LAW. SOME COUNTRIES OR STATES DO NOT ALLOW THE EXCLUSION OR LIMITATION OF LIABILITY FOR CONSEQUENTIAL, SPECIAL, INDIRECT, INCIDENTAL DAMAGES AND, ACCORDINGLY, SOME PORTIONS OF THESE LIMITATIONS MAY NOT APPLY TO YOU. BY YOUR OPENING OF THIS SEALED PACKAGE OR INSTALLING OR USING THE SOFTWARE, YOU HAVE ACCEPTED THAT THE ABOVE LIMITATIONS OR THE MAXIMUM LEGALLY APPLICABLE SUBSET OF THESE LIMITATIONS APPLY TO YOUR PURCHASE OF THIS SOFTWARE.'

### **Printing History**

December 2006  
Edition 2007





*Phi Software*

## **CONTENTS**



|   |           |
|---|-----------|
| <b>MIKE SHE - Getting Started</b>                     | <b>15</b> |
| 1 INTRODUCTION  | 17        |
| 1.1 Service and Maintenance                           | 20        |
| 1.1.1 Service Packs                                   | 21        |
| 2 MIKE SHE  | 23        |
| 2.1 Getting Help                                      | 23        |
| 2.2 Hardware Requirements                             | 23        |
| 2.3 MIKE Zero   | 25        |
| 2.3.1 Creating a new MIKE SHE Project                 | 26        |
| 2.3.2 Creating a New Model File                       | 27        |
| 2.3.3 Opening an Existing Model File                  | 29        |
| 2.4 Importing Existing Release 2005 Projects          | 30        |
| 2.4.1 Linking 2005 Results in a 2007 .she Set Up File | 31        |
| 2.5 MIKE Zero Editors                                 | 32        |
| 2.6 The MIKE SHE User Interface                       | 33        |
| 2.6.1 The Setup Data Tree                             | 35        |
| 3 BUILDING A MIKE SHE MODEL                           | 37        |
| 3.1 Adding Background Maps                            | 37        |
| 3.2 Selecting the Processes                           | 37        |
| 3.3 Setting up the Simulation parameters              | 38        |
| 3.3.1 Hot Starting from a previous simulation         | 39        |
| 3.4 Defining the model domain and grid                | 39        |
| 3.4.1 MIKE SHE model limits                           | 41        |
| 3.4.2 MIKE SHE Demo model limits                      | 42        |
| 3.5 Defining Topography                               | 42        |
| 3.6 Adding Precipitation                              | 43        |
| 3.7 Adding Surface Water                              | 44        |
| 3.8 Adding Overland Flow                              | 44        |
| 3.8.1 Overland Flow Boundary Conditions               | 45        |
| 3.9 Adding Unsaturated Flow                           | 45        |
| 3.9.1 UZ Classification                               | 45        |
| 3.9.2 Coupling Between Unsaturated and Saturated Zone | 46        |
| 3.10 Adding Groundwater                               | 47        |
| 3.10.1 Your Conceptual Geologic Model                 | 47        |
| 3.10.2 Working with Lenses                            | 50        |
| 3.10.3 Numerical Layers                               | 51        |
| 3.10.4 Groundwater Drainage                           | 51        |
| 3.10.5 MIKE SHE versus MODFLOW                        | 55        |
| 3.10.6 Importing a MODFLOW 96 or MODFLOW 2000 Model   | 57        |



|        |   |           |
|--------|---|-----------|
| 3.11   | Setting up your results . . . . .                                 | 59        |
| 3.11.1 | Importing an ASCII file for Detailed Time Series Output . . . . . | 59        |
| 4      | RUNNING YOUR MIKE SHE MODEL . . . . .                             | 61        |
| 4.1    | Preprocessing your model . . . . .                                | 61        |
| 4.1.1  | Loading your Preprocessed Data . . . . .                          | 62        |
| 4.1.2  | Exporting, Saving and Editing the pre-processed data . . . . .    | 62        |
| 4.1.3  | Updating Overlays in Pre-processed views . . . . .                | 63        |
| 4.1.4  | Pre-processed Data Views . . . . .                                | 63        |
| 4.2    | Running your Model . . . . .                                      | 68        |
| 4.2.1  | Running MIKE SHE from a Batch File . . . . .                      | 68        |
| 4.2.2  | Controlling the Time Steps . . . . .                              | 70        |
| 4.2.3  | Speeding up your simulation . . . . .                             | 73        |
|        | <b>Working with your Results . . . . .</b>                        | <b>75</b> |
| 5      | THE RESULTS TAB . . . . .   | 77        |
| 5.1    | Linking 2005 Results in a 2007 .she Set Up File . . . . .         | 77        |
| 5.2    | Detailed Time Series Results . . . . .                            | 78        |
| 5.3    | Gridded Results . . . . .   | 79        |
| 5.3.1  | “The Result Viewer setup file already exists” warning . . . . .   | 80        |
| 5.4    | MIKE 11 Detailed Time Series . . . . .                            | 81        |
| 5.5    | Run Statistics . . . . .  | 82        |
| 5.5.1  | Shape file output for run statistics . . . . .                    | 83        |
| 6      | OUTPUT ITEMS . . . . .  | 85        |
| 7      | THE RESULTS VIEWER . . . . .                                      | 93        |
| 7.1    | Toolbars . . . . .  | 93        |
| 7.2    | Modifying the plot . . . . .                                      | 95        |
| 7.2.1  | Adding additional result files and overlays . . . . .             | 95        |
| 7.2.2  | Adding or modifying vectors . . . . .                             | 98        |
| 7.2.3  | Changing the shading or contour settings . . . . .                | 100       |
| 7.2.4  | Changing the legend and colour scale . . . . .                    | 105       |
| 7.3    | Displaying a time series at a point . . . . .                     | 108       |
| 7.4    | Saturated Zone Cross-section Plots . . . . .                      | 111       |
| 7.4.1  | Changing to a different SZ layer . . . . .                        | 115       |
| 7.5    | Displaying a MIKE 11 cross-section . . . . .                      | 115       |
| 7.6    | UZ Specific Plots . . . . .                                       | 116       |
| 7.6.1  | UZ Scatter and Filled Plots . . . . .                             | 116       |
| 7.6.2  | UZ Plot . . . . .   | 117       |



|        |   |     |
|--------|---|-----|
| 8      | USING THE WATER BALANCE TOOL                            | 121 |
| 8.1    | Creating a water balance                                | 121 |
| 8.1.1  | Create a new water balance document                     | 122 |
| 8.1.2  | Extract the water balance data                          | 123 |
| 8.1.3  | Specify your water balance                              | 125 |
| 8.1.4  | Calculate and View the Water Balance                    | 129 |
| 8.2    | Calculating Water Balances in Batch Mode                | 129 |
| 8.3    | Available Water Balance Items                           | 130 |
| 8.4    | Standard Water Balance Types                            | 136 |
| 8.5    | Making Custom Water Balances                            | 139 |
| 8.6    | Water Balance Restrictions                              | 140 |
| 8.6.1  | Linear Reservoir and Simple Sub-catchment Overland Flow | 140 |
|        | <b>Additional Options</b>                               | 141 |
| 9      | EXTRA PARAMETERS  | 143 |
| 9.1    | Sheet Pile Module                                       | 143 |
| 9.1.1  | Sheet Pile Location                                     | 144 |
| 9.1.2  | Leakage Coefficient                                     | 144 |
| 9.1.3  | Top and bottom levels (optional)                        | 145 |
| 9.1.4  | Input File for the Sheet Pile Module                    | 145 |
| 9.2    | Negative Precipitation                                  | 148 |
| 9.3    | Precipitation Multiplier                                | 149 |
| 9.4    | SZ Drainage to Specified MIKE 11 H-points               | 149 |
| 9.5    | Time varying drainage parameters                        | 152 |
| 9.6    | Irrigation River Source Factors                         | 153 |
| 9.7    | Distributed Snow Melt Constants                         | 155 |
| 9.8    | Canyon exchange option for deep narrow channels         | 155 |
| 9.9    | Simplified Overland Flow Options                        | 156 |
| 9.10   | 2-Layer UZ Options                                      | 157 |
| 9.10.1 | Threshold depth for infiltration                        | 157 |
| 9.10.2 | Increase infiltration to dry soils                      | 158 |
| 9.11   | GeoViewer Output  | 158 |
|        | <b>Surface Water Modelling with MIKE 11</b>             | 161 |
| 10     | COUPLING MIKE 11 AND MIKE SHE                           | 163 |
| 10.1   | MIKE SHE  | 163 |
| 10.2   | MIKE 11   | 164 |
| 10.2.1 | MIKE 11 Cross-sections                                  | 164 |
| 10.2.2 | MIKE SHE Coupling Reaches                               | 167 |





|  |  |            |
|--|--|------------|
| 10.3   | Converting from Flood Codes to Overbank Spilling . . . . .   | 174        |
| 10.4   | Common MIKE 11 Error Messages . . . . .  | 175        |
| 10.4.1   | Error No 25: At the h-point ____ the water depth greater than 4 times<br>max. depth                  | 175        |
| 10.4.2   | Warning No 47: At the h-point ____ the water level as fallen below<br>the bottom of the slot x times | 175        |
| <b>Drainage modelling with MOUSE . . . . .</b> |  | <b>177</b> |
| 11   | USING MIKE SHE WITH MOUSE . . . . .  | 179        |
| 11.1   | Coupling MIKE SHE and MOUSE . . . . .  | 182        |
| 11.1.1   | Telling MIKE SHE to couple to MOUSE . . . . .  | 183        |
| 11.1.2   | Telling MOUSE that it is coupled to a MIKE SHE model . . . . .                                       | 183        |
| 11.1.3   | Creating a MsheMouse.pfs file . . . . .  | 184        |
| 11.1.4   | Output Files . . . . .   | 186        |
| 11.2   | Running the Coupled Models . . . . .   | 187        |
| 11.2.1   | Warning messages . . . . .   | 187        |
| 11.2.2   | Water Balance Limitation . . . . .   | 188        |
| <b>Working with Water Quality . . . . .</b>    |  | <b>189</b> |
| 12   | SIMULATING WATER QUALITY . . . . .   | 191        |
| 12.1   | Flow Storing Requirements . . . . .  | 192        |
| 12.2   | Storing of Results . . . . .   | 192        |
| 12.3   | Simulation and Time Step Control . . . . .   | 193        |
| 12.3.1   | Calibrating and Verifying the Model . . . . .  | 193        |
| 13   | USING THE FULLY INTEGRATED AD MODULE . . . . .   | 195        |
| 13.1   | Data Units . . . . .   | 195        |
| 13.2   | Executing MIKE SHE AD . . . . .  | 195        |
| 13.3   | Working with the .TSF Files . . . . .  | 196        |
| 13.3.1   | Species INDEPENDENT Data . . . . .   | 199        |
| 13.3.2   | Species DEPENDENT Data . . . . .   | 206        |
| 13.4   | Working with the .XTSF File . . . . .  | 212        |
| 13.4.1   | Simulation Parameters . . . . .  | 214        |
| 13.5   | Example input files . . . . .  | 219        |
| 13.5.1   | Transport Setup File (*.tsf) . . . . .   | 219        |
| 13.5.2   | Extra transport setup file (.xtsf) . . . . .   | 223        |
| 14   | WORKING WITH PARTICLE TRACKING . . . . .   | 227        |
| 14.1   | Requirements in MIKE SHE WM . . . . .  | 227        |
| 14.1.1   | Flow Storing Requirements . . . . .  | 228        |



|                                 |   |     |
|---------------------------------|---|-----|
| 14.1.2                          | Specification of Well Fields              | 228 |
| 14.1.3                          | Input to the PT Simulations               | 228 |
| 14.2                            | Executing MIKE SHE PT                     | 232 |
| 14.3                            | Output from the PT simulations            | 232 |
| 14.4                            | PTBin Output Retrieval Utility            | 233 |
| 14.5                            | PT Output Retrieval Utility               | 234 |
| 14.5.1                          | Limitations with the PT Output Retrieval  | 235 |
| 14.5.2                          | Structure of the PT_OR File               | 235 |
| 14.5.3                          | Example PT Output Retrieval File (.PT_OR) | 236 |
| <b>Working with Time Series</b> |   | 239 |
| 15                              | TIME SERIES DATA                          | 241 |
| 15.1                            | Creating Time Series in MIKE SHE          | 241 |
| 15.1.1                          | Import from ASCII                         | 242 |
| 15.1.2                          | Import from Excel                         | 243 |
| 15.1.3                          | Import from old .t0 file                  | 243 |
| 15.2                            | Working with Spatial Time Series          | 243 |
| 15.3                            | Time Series Types                         | 244 |
| <b>Time Series Editor</b>       |   | 247 |
| 16                              | INTRODUCTION                              | 249 |
| 17                              | PROPERTIES                                | 251 |
| 17.1                            | New File Dialog                           | 251 |
| 17.1.1                          | Wave Climate Template                     | 252 |
| 17.1.2                          | LITProf template                          | 253 |
| 17.1.3                          | LITTren template                          | 253 |
| 17.1.4                          | Source Template                           | 254 |
| 17.1.5                          | STPBatch template                         | 255 |
| 17.1.6                          | Wind template                             | 256 |
| 17.2                            | File Properties Dialog                    | 256 |
| 17.2.1                          | General Information                       | 257 |
| 17.2.2                          | Axis Information                          | 257 |
| 17.2.3                          | Item Information                          | 258 |
| 17.3                            | Tabular View                              | 260 |
| 17.4                            | Graphical View                            | 260 |
| 17.4.1                          | Zoom                                      | 261 |
| 17.4.2                          | Editing modes                             | 261 |
| 17.4.3                          | Graphical and font settings               | 262 |
| 17.4.4                          | TS Types graphical representation         | 262 |



|      |   |            |
|------|---|------------|
| 17.5 | Graphical Settings Dialog . . . . .                       | 267        |
| 17.6 | Font Settings Dialog . . . . .                            | 268        |
| 17.7 | File Formats . . . . .                                    | 269        |
| 17.8 | Import from ASCII File . . . . .                          | 270        |
| 17.9 | Export to ASCII File . . . . .                            | 273        |
| 18   | TOOLS . . . . .   | 275        |
| 18.1 | Calculator . . . . .                                      | 275        |
| 18.2 | Interpolation . . . . .                                   | 276        |
| 18.3 | Select Sub-Set . . . . .                                  | 277        |
| 18.4 | Statistics . . . . .                                      | 278        |
|      | <b>Working with Spatial Data . . . . .</b>                | <b>281</b> |
| 19   | SPATIAL DATA IN MIKE SHE . . . . .                        | 283        |
| 19.1 | Using MIKE SHE with ArcGIS . . . . .                      | 283        |
| 19.2 | Integer Grid Codes . . . . .                              | 284        |
| 19.3 | Gridded (.dfs2) Data . . . . .                            | 286        |
|      | 19.3.1 Stationary Real Data . . . . .                     | 287        |
|      | 19.3.2 Time-varying Real Data . . . . .                   | 288        |
|      | 19.3.3 Integer Grid Codes . . . . .                       | 290        |
| 19.4 | Interpolation Methods . . . . .                           | 291        |
|      | 19.4.1 Bilinear Interpolation . . . . .                   | 292        |
|      | 19.4.2 Triangular Interpolation . . . . .                 | 296        |
| 19.5 | Node numbering in the Grid Editor . . . . .               | 297        |
| 19.6 | Performing simple math on multiple grids . . . . .        | 297        |
| 19.7 | Performing complex operations on multiple grids . . . . . | 300        |
|      | <b>Grid Editor . . . . .</b>                              | <b>301</b> |
| 20   | INTRODUCTION . . . . .                                    | 303        |
| 20.1 | Create a New Dataset . . . . .                            | 303        |
| 20.2 | Open an Existing Dataset . . . . .                        | 304        |
| 20.3 | Editing the Dataset . . . . .                             | 304        |
| 20.4 | Further help . . . . .                                    | 304        |
| 21   | FILE . . . . .  | 307        |
| 21.1 | File Formats . . . . .                                    | 307        |
|      | 21.1.1 DFS format . . . . .                               | 307        |
|      | 21.1.2 ASCII format . . . . .                             | 307        |
|      | 21.1.3 Grid State Format . . . . .                        | 308        |
| 21.2 | New File . . . . .  | 309        |



|        |   |     |
|--------|---|-----|
| 21.2.1 | Step 1: Select the type of grid to be created . . . . .   | 309 |
| 21.2.2 | Step 2: Specify the projection, the geographical position of the origin<br>of the dataset and the orientation | 309 |
| 21.2.3 | Step 3: Specify the temporal and spatial properties . . . . .   | 311 |
| 21.2.4 | Step 4: Specify the items to be included . . . . .  | 311 |
| 21.2.5 | Step 5: Overview . . . . .  | 311 |
| 21.3   | Import from ASCII File . . . . .  | 311 |
| 21.3.1 | File to Import . . . . .  | 312 |
| 21.3.2 | Completion and Editing . . . . .  | 312 |
| 21.3.3 | Hint . . . . .  | 312 |
| 21.4   | Export to ASCII File . . . . .  | 312 |
| 21.5   | Import from Dfsu File . . . . .   | 312 |
| 21.5.1 | Step 1: Select File to Import . . . . .   | 312 |
| 21.5.2 | Step 2: Specify geographical parameters . . . . .   | 312 |
| 21.5.3 | Step 3: Specify the spatial properties . . . . .  | 313 |
| 21.5.4 | Step 4: Specify land value . . . . .  | 313 |
| 21.5.5 | Step 5: Completion and Editing . . . . .  | 313 |
| 22     | EDIT . . . . .  | 315 |
| 22.1   | Geographical Information . . . . .  | 315 |
| 22.2   | Time Steps . . . . .  | 317 |
| 22.2.1 | Modifying Time Steps . . . . .  | 317 |
| 22.3   | Items . . . . .   | 318 |
| 22.3.1 | Editing an existing file . . . . .  | 319 |
| 22.4   | Spatial Axis . . . . .  | 319 |
| 22.5   | Edit Custom Blocks . . . . .  | 319 |
| 22.5.1 | MIKE 21 / MIKE 3 specific . . . . .   | 320 |
| 23     | VIEW . . . . .  | 321 |
| 23.1   | Tabular View . . . . .  | 321 |
| 23.2   | Graphical View . . . . .  | 322 |
| 23.3   | Palette . . . . .   | 322 |
| 23.3.1 | Palette Wizard, step 1 of 3 . . . . .   | 323 |
| 23.3.2 | Palette Wizard, step 2 of 3 . . . . .   | 324 |
| 23.3.3 | Palette Wizard, step 3 of 3 . . . . .   | 325 |
| 23.3.4 | Open . . . . .  | 325 |
| 23.3.5 | Save . . . . .  | 326 |
| 23.4   | Overlay . . . . .   | 326 |
| 23.5   | Grid Settings . . . . .   | 327 |
| 23.5.1 | Contour type . . . . .  | 327 |
| 23.5.2 | Isolines . . . . .  | 327 |
| 23.6   | Axis Annotation . . . . .   | 328 |



|       |   |            |
|-------|---|------------|
| 23.7  | Mouse Pointer Coordinates . . . . .                         | 328        |
| 23.8  | Fixed aspect ratio . . . . .                                | 328        |
| 23.9  | Export Graphics . . . . .                                   | 328        |
| 23.10 | Toolbars . . . . .  | 328        |
|       | 23.10.1 Grid Editor Tools . . . . .                         | 329        |
|       | 23.10.2 Grid Editor Navigation . . . . .                    | 329        |
| 23.11 | Status Bar . . . . .  | 330        |
| 24    | TOOLS . . . . .   | 331        |
| 24.1  | Navigation . . . . .  | 331        |
| 24.2  | Go to . . . . .   | 331        |
| 24.3  | Synchronize . . . . .                                       | 331        |
| 24.4  | Selection . . . . .   | 332        |
|       | 24.4.1 Select and deselect . . . . .                        | 332        |
|       | 24.4.2 Select a Sub-Set of Data . . . . .                   | 333        |
| 24.5  | Interpolation . . . . .                                     | 334        |
|       | 24.5.1 Active Dataset . . . . .                             | 334        |
|       | 24.5.2 Interpolation Settings . . . . .                     | 334        |
|       | 24.5.3 Search Type . . . . .                                | 334        |
| 24.6  | Filter . . . . .  | 335        |
| 24.7  | Set Value . . . . .   | 335        |
| 24.8  | Calculator . . . . .  | 336        |
|       | 24.8.1 List of Functions . . . . .                          | 336        |
| 24.9  | Calculate Statistics . . . . .                              | 339        |
| 24.10 | Copy File into Data . . . . .                               | 340        |
|       | 24.10.1 File to Copy . . . . .                              | 340        |
|       | 24.10.2 Item Mapping . . . . .                              | 340        |
|       | 24.10.3 2D to 3D Layer Mapping . . . . .                    | 341        |
|       | 24.10.4 Sub-area Position . . . . .                         | 341        |
|       | 24.10.5 Time Position . . . . .                             | 341        |
|       | 24.10.6 Operation . . . . .                                 | 341        |
| 24.11 | Crop . . . . .  | 341        |
| 25    | DATA OVERLAY . . . . .                                      | 343        |
| 25.1  | Image Manager . . . . .                                     | 343        |
| 25.2  | Overlay Manager . . . . .                                   | 345        |
|       | <b>MIKE ZERO Options . . . . .</b>                          | <b>347</b> |
| 26    | EUM DATA UNITS . . . . .                                    | 349        |
| 26.1  | Changing from SI to Imperial (American) data units. . . . . | 351        |
| 26.2  | Restoring the default units . . . . .                       | 352        |



|                |  |            |
|----------------|--|------------|
| 26.3           | Changing the EUM data type of a Parameter . . . . .  | 352        |
| 26.3.1         | Changing the EUM Type of a .dfs0 Parameter . . . . . | 354        |
| 26.3.2         | Changing the EUM Type of a .dfs2 Parameter . . . . . | 355        |
| <b>AUTOCAL</b> | . . . . .  | <b>357</b> |
| 27             | AUTO CALIBRATION TOOL . . . . .                      | 359        |
| 27.1           | New AUTOCAL Dialog . . . . .                         | 360        |
| 27.2           | Simulation Specifications . . . . .                  | 361        |
| 27.2.1         | Model simulation sequence . . . . .                  | 362        |
| 27.2.2         | Model parameter files . . . . .                      | 364        |
| 27.2.3         | Simulation option . . . . .                          | 367        |
| 27.2.4         | Simulation title . . . . .                           | 367        |
| 27.3           | Model Parameters . . . . .                           | 367        |
| 27.4           | Objective Functions . . . . .                        | 370        |
| 27.4.1         | Output measures . . . . .                            | 371        |
| 27.4.2         | Objective functions . . . . .                        | 373        |
| 27.4.3         | Evaluation period . . . . .                          | 374        |
| 27.4.4         | Aggregation of objective functions . . . . .         | 374        |
| 27.5           | Scenario Runs . . . . .                              | 376        |
| 27.5.1         | Scenario type . . . . .                              | 376        |
| 27.5.2         | Parameters . . . . .                                 | 376        |
| 27.6           | Sensitivity Analysis . . . . .                       | 377        |
| 27.6.1         | Sensitivity analysis method . . . . .                | 377        |
| 27.6.2         | Local sensitivity analysis . . . . .                 | 377        |
| 27.7           | Parameter Optimisation . . . . .                     | 379        |
| 27.7.1         | Optimisation method . . . . .                        | 380        |
| 27.7.2         | Shuffled Complex Evolution method . . . . .          | 380        |
| 27.7.3         | Population Simplex Evolution method . . . . .        | 384        |
| 27.8           | Save Output Files . . . . .                          | 386        |
| 27.9           | Office Grid . . . . .                                | 387        |
| 27.10          | Start AUTOCAL Simulation . . . . .                   | 389        |
| 27.11          | Output . . . . .                                     | 389        |
| 27.11.1        | General output files . . . . .                       | 389        |
| 27.11.2        | Sensitivity analysis output . . . . .                | 390        |
| 27.11.3        | Optimisation output . . . . .                        | 391        |
| 27.12          | Error handling . . . . .                             | 391        |
| <b>Index</b>   | . . . . .  | <b>393</b> |



*Phi Software*

**MIKE SHE - GETTING STARTED**







## 1 INTRODUCTION

In the hydrological cycle, water evaporates from the oceans, lakes and rivers, from the soil and is transpired by plants. This water vapour is transported in the atmosphere and falls back to the earth as rain and snow. It infiltrates to the groundwater and discharges to streams and rivers as base-flow. It also runs off directly to streams and rivers that flow back to the ocean. The hydrologic cycle is a closed loop and our interventions do not remove water; rather they affect the movement and transfer of water within the hydrologic cycle.

In 1969, Freeze and Harlan (Freeze and Harlan, 1969) proposed a blueprint for modelling the hydrologic cycle. In this original blueprint, different flow processes were described by their governing partial differential equations. The equations used in the blueprint were known to represent the physical processes at the appropriate scales in the different parts of the hydrological cycle.

From 1977 onwards, a consortium of three European organizations developed, and extensively applied, the *Système Hydrologique Européen* (SHE) based on the blueprint of Freeze and Harlan (Abbott et al., 1986a & b). The integrated hydrological modelling system, MIKE SHE, emerged from this work (see Figure 1.1)

Since the mid-1980's, MIKE SHE has been further developed and extended by DHI Water & Environment. Today, MIKE SHE is an advanced, flexible framework for hydrologic modelling. It includes a full suite of pre- and post-processing tools, plus a flexible mix of advanced and simple solution techniques for each of the hydrologic processes. MIKE SHE covers the major processes in the hydrologic cycle and includes process models for evapotranspiration, overland flow, unsaturated flow, groundwater flow, and channel flow and their interactions. Each of these processes can be represented at different levels of spatial distribution and complexity, according to the goals of the modelling study, the availability of field data and the modeller's choices, (Butts et al. 2004). The MIKE SHE user interface allows the user to intuitively build the model description based on the user's conceptual model of the watershed. The model data is specified in a variety of formats independent of the model domain and grid, including native GIS formats. At run time, the spatial data is mapped onto the numerical grid, which makes it easy to change the spatial discretisation.

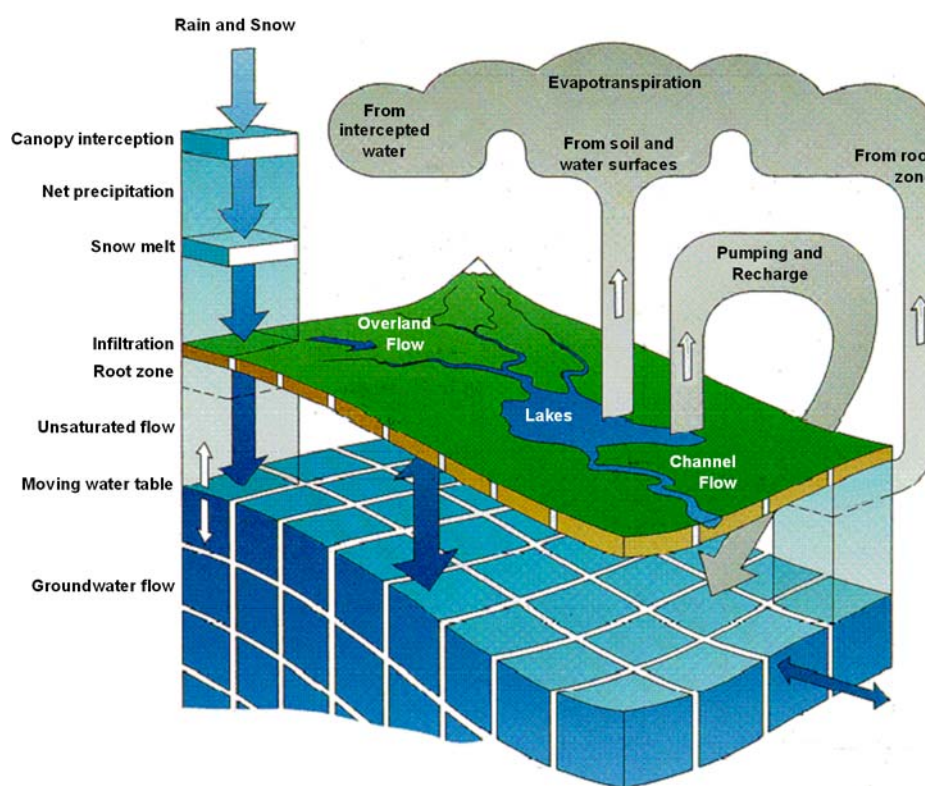


Figure 1.1 .Hydrologic processes simulated by MIKE SHE

MIKE SHE uses MIKE 11 to simulate channel flow. MIKE 11 includes comprehensive facilities for modelling complex channel networks, lakes and reservoirs, and river structures, such as gates, sluices, and weirs. In many highly managed river systems, accurate representation of the river structures and their operation rules is essential. In a similar manner, MIKE SHE is also linked to the MOUSE sewer model, which can be used to simulate the interaction between urban storm water and sanitary sewer networks and groundwater. MIKE SHE is applicable at spatial scales ranging from a single soil profile, for evaluating crop water requirements, to large regions including several river catchments, such as the 80,000 km<sup>2</sup> Senegal Basin (e.g. Andersen et al., 2001). MIKE SHE has proven valuable in hundreds of research and consultancy projects covering a wide range of climatological and hydrological regimes, many of which are referenced in Graham and Butts (2006).

The need for fully integrated surface and groundwater models, like MIKE SHE, has been highlighted by several recent studies (e.g. Camp Dresser &



McKee Inc., 2001; Kaiser-Hill, 2001; West Consultants Inc. et al., 2001; Kimbley-Horn & Assoc. Inc. et al., 2002; Middlemis, 2004, which can all be downloaded from the MIKE SHE web site). These studies compare and contrast available integrated groundwater/surface water codes. They also show that few codes exist that have been designed and developed to fully integrate surface water and groundwater. Further, few of these have been applied outside of the academic community (Kaiser-Hill, 2001).

### **Applications around the world**

MIKE SHE has been used in a broad range of applications. It is being used operationally in many countries around the world by organizations ranging from universities and research centres to consulting engineers companies (Refsgaard & Storm, 1995). MIKE SHE has been used for the analysis, planning and management of a wide range of water resources and environmental and ecological problems related to surface water and groundwater, such as:

- River basin management and planning
- Water supply design, management and optimization
- Irrigation and drainage
- Soil and water management
- Surface water impact from groundwater withdrawal
- Conjunctive use of groundwater and surface water
- Wetland management and restoration
- Ecological evaluations
- Groundwater management
- Environmental impact assessments
- Aquifer vulnerability mapping
- Contamination from waste disposal
- Surface water and groundwater quality remediation
- Floodplain studies
- Impact of land use and climate change
- Impact of agriculture (irrigation, drainage, nutrients and pesticides, etc.)

Graham and Butts (2006) contains a list of some easily accessible references for many of the application areas listed above.



## **User interface**

MIKE SHE's user interface can be characterized by the need to

- 1 Develop a GUI that promotes a logical and intuitive workflow, which is why it includes
  - A dynamic navigation tree that depends on simple and logical choices
  - A conceptual model approach that is translated at run-time into the mathematical model
  - Object oriented “thinking” (geo-objects with attached properties)
  - Full, context-sensitive, on-line help
  - Customized input/output units to support local needs
- 2 Strengthen the calibration and result analysis processes, which is why it includes
  - Default HTML outputs (calibration hydrographs, goodness of fit, water balances, etc.)
  - User-defined HTML outputs
  - A Result Viewer that integrates 1D, 2D and 3D data for viewing and animation
  - Water balance, auto-calibration and parameter estimation tools.
- 3 Develop a flexible, unstructured GUI suitable for different modelling approaches, which is why it includes
  - Flexible data format (gridded data, .shp files, etc.) that is easy to update for new data formats
  - Flexible time series module for manipulating time-varying data
  - Flexible engine structure that can be easily updated with new numerical engines

The result is a GUI that is flexible enough for the most complex applications imaginable, yet remains easy-to-use for simple applications.

## **1.1 Service and Maintenance**

As with any complex software package, the software is being continually improved and extended. Some of these improvements are fixes of problems that have slipped through our quality control. Others are fixes of known minor problems with the software. However, the vast majority of



the changes in new releases and service packs are related to improvements to the functionality of the software.

Your initial purchase of the software is protected by a one-year subscription to our Service and Maintenance Agreement. Your Service and Maintenance Agreement entitles you free support for software problems via email or telephone and regular updates to the software.

We strongly recommend that you subscribe to the Service and Maintenance Agreement after the first year to further protect your investment. Improvements, extensions and fixes are continually being made, and we will make every effort to help you with any problems that you encounter, but we cannot provide fixes for any versions older than the current release.

### **1.1.1 Service Packs**

As part of the Service and Maintenance, there is an auto update program installed with your software. This program automatically checks our website for Service Packs to the currently installed release and downloads the Service Pack if it is available. You will be asked before the installation begins, if you want the installation to proceed. We strongly recommend that you install the latest Service Pack as soon as they are released.

However, some clients prefer not to install the Service Pack during a project, or close to the end of a project. Occasionally, a fix in the numerical engine will slightly change your simulation results. This may require you to re-run previously finished simulations to obtain valid comparisons between simulations.

The Auto Updater overwrites your existing executable files. Therefore, if you are concerned about potential changes in your results, they you should backup all of the files in the MIKE SHE installation directory, before installing the Service Pack.

If you did not back up your installation directory, and you need to restore a previous version, DHI maintains an archive of all standard patch versions. Contact your local support centre and we will send you a copy of your previous executable.





## 2 **MIKE SHE**

The MIKE SHE is part of the MIKE Zero suite of modelling tools, which is a global user interface for managing and manipulating data files and projects for many of the DHI Software products. Thus, when you launch MIKE SHE or one of the other MIKE Zero products you are really launching MIKE Zero. MIKE Zero then provides the framework from which you can run MIKE SHE, or any other product in the MIKE Zero family.

### 2.1 **Getting Help**

If you click F1 in any MIKE SHE dialogue, you will land in one of the sections of The MIKE SHE Reference Guide. Likewise, if you click F1 in any MIKE 11 or other MIKE Zero dialogue, you will land in a relevant section of the on-line help.

This manual, the MIKE SHE - Getting Started manual, is a supplement to the basic on-line F1 help and provides you with additional information on how to use MIKE SHE to get the results that you want.

### 2.2 **Hardware Requirements**

The hardware requirements for MIKE SHE depend on the model that you are trying to simulate. As a rule of thumb, any good quality, new computer should be sufficient for an average MIKE SHE model. Thus, a typical machine for running an average MIKE SHE model will have at least a 1.8 to 2 GHz CPU, 1 to 2GB of RAM, and 30 to 50 GB of free disk space.

**Note:** MIKE SHE will not run on Linux workstations.

#### **64-bit CPU**

DHI has been working with the Microsoft X64 team and some of the newer DHI numerical engines are compiled for a 64-bit processor. However, MIKE 11 is an older code that cannot yet be compiled for a 64-bit processor. MIKE 11, and thus also MIKE SHE, will run fine on a 64-bit machine, but it will run as a 32-bit application.

#### **Dual Core / Dual Processor / Hyper Threading**

A dual core or dual processor machine will allow you to run MIKE SHE and a second application, such as a spreadsheet. However, the MIKE SHE code has not been optimized for parallel processing, so the MIKE SHE engine cannot take advantage of the parallelization possibilities.



If you are using the AUTOCAL program, you might want to consider the Office Grid package. This package will allow you to spread multiple simulations out over an office network.

On the other hand, a dual core or dual processor machine will allow you to work on something else, while your simulation runs.

### **RAM**

MIKE SHE does not dynamically allocate RAM. This means that a large model will allocate a lot of RAM even if the cells are not being used. Thus to run a large model, you will need the most RAM possible. If you don't have enough RAM, then MIKE SHE will swap to the hard disk, which can slow down your simulation by a factor of 10 or more.

However, unlike the newer 64-bit Windows operating system, the 32-bit Windows operating system is restricted to 2GB of RAM. Thus, if you have a 64-bit system and even if you have more than 2GB of RAM, MIKE SHE will only use a maximum of 2GB. The 2GB limit is for each application, so if you are running multiple applications, then extra RAM can be useful.

### **CPU Speed**

Since, you cannot get any speed advantage from a dual processor or a 64-bit processor, then the only way to speed up your simulation is to get the fastest (with respect to clock speed) processor possible. This is typically not the newest processors, since the newer processors are being designed for 64-bits and dual configurations. In principle, you can overclock your processor, but this has some heat implications. So, you should talk to your computer vendor about your options.

### **Keep it simple!**

In most cases, the best way to speed up your model is to make it simpler. You should look very carefully at your model and ask yourself the following questions, for example:

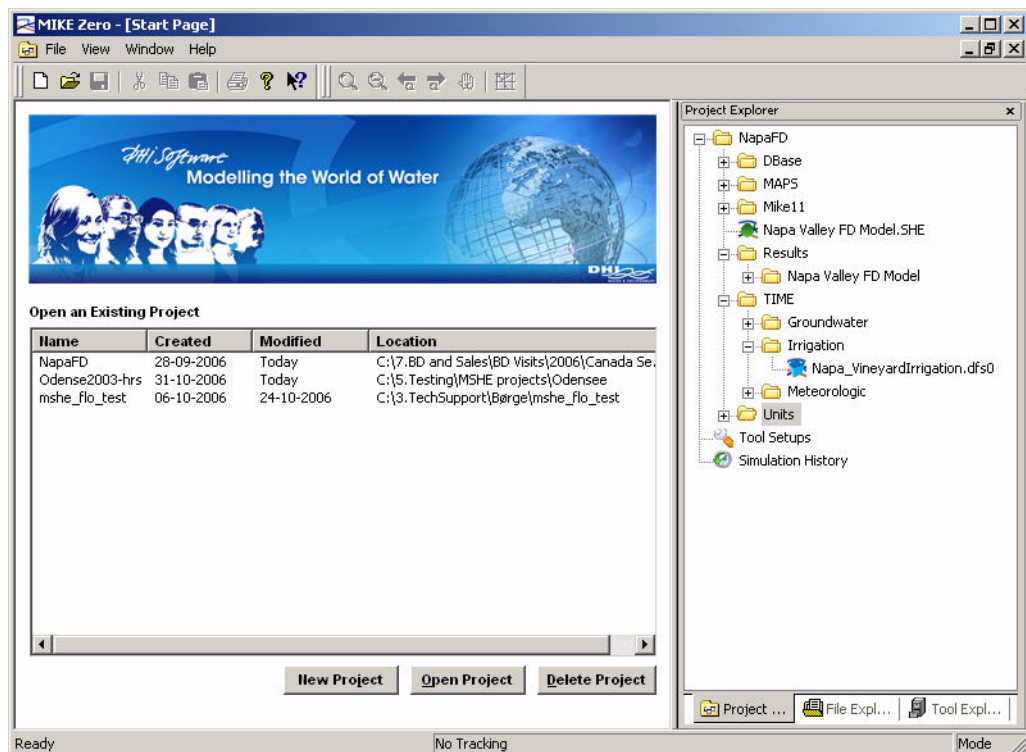
- **Do you really need a fine discretisation during calibration?** - A coarser grid may allow you to do many more calibration runs. Then when the model is calibrated, you can refine the grid for the final simulations - but remember to check your calibration first.





- **Do you really need the Richards equation for unsaturated flow?** - For regional models, the two layer water balance method is usually sufficient, which is very fast. The gravity flow method is also, typically 2-5 times faster than the Richards equation method. Again during the calibration it can be a good idea to use one of the simpler methods and the more detailed method for the final simulations.
- **Is your MIKE 11 simulation too detailed?** - If your MIKE 11 cross-sections are too close together, MIKE 11 will run with a very short time step. Regional models can often be run with the simple routing methods in MIKE 11, which are very fast.

## 2.3 MIKE Zero



MIKE Zero is more than a set of modelling tools. MIKE Zero is a project management interface, with a full range of tools for helping you with your modelling project.

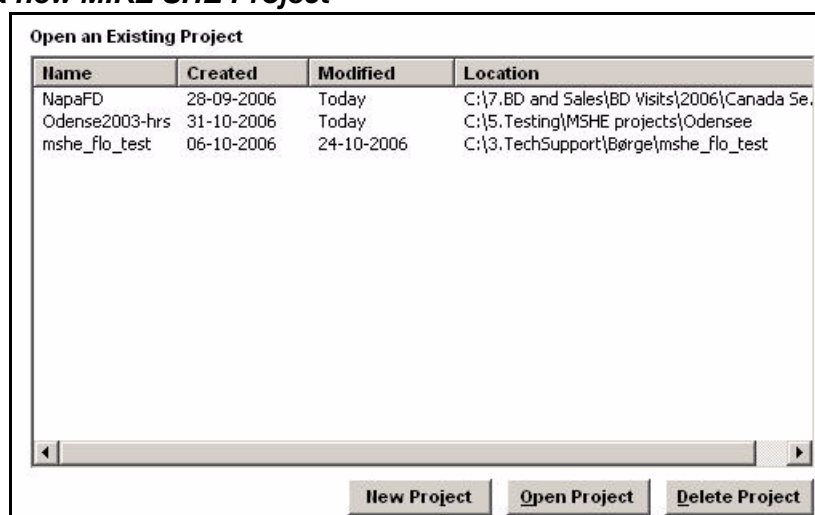
The MIKE Zero Start Page (see above) includes three main components, , the Project Overview table, the Project Explorer data tree, and the top Menu and Icon bar.



A MIKE Zero project includes all of the modelling artifacts; that is all of the raw data files, model input files, and model output files, as well as any reports, spreadsheets, plots, etc. In any project, it is a challenge to maintain an overview of all of these files, not to mention keeping regular backups and archives of all of these files. As you progress through the calibration and validation phases, and then on to the scenario analysis and report writing phases, the number of model artifacts can become overwhelming.

The MIKE Zero project structure is designed to help you keep control of your project.

### 2.3.1 Creating a new MIKE SHE Project



| Name           | Created    | Modified   | Location                                    |
|----------------|------------|------------|---|
| NapaFD         | 28-09-2006 | Today      | C:\7.BD and Sales\BD Visits\2006\Canada Se. |
| Odense2003-hrs | 31-10-2006 | Today      | C:\5.Testing\MSHE projects\Odensee          |
| mshe_flo_test  | 06-10-2006 | 24-10-2006 | C:\3.TechSupport\Børge\mshe_flo_test        |

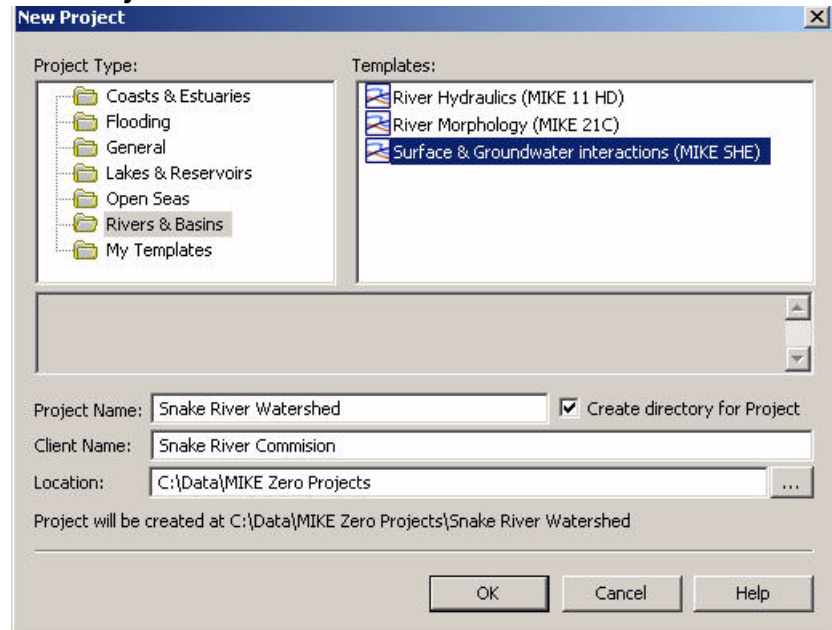
New Project Open Project Delete Project

The Project Overview table lists the projects that you have created, along with their creation and last modified dates.

The core of each project is a .mzp file, which contains all of the references to all of the files. The Location listed is where the *projectname.mzp* file is located.



### New Project button



The easiest way to create a new MIKE SHE project is to use the New Project Button, which will display the above dialogue.


When you create a new MIKE SHE project with the New Project button, a series of project templates are available. For MIKE SHE, you can use the **Surface and Groundwater Interaction** template under the **Rivers and Basins** section.

Typically, each project is stored in one directory, with a number of subdirectories. The template describes the default organization of the directory and its subdirectories. When you select OK, a new empty project directory structure will be created.

### Delete Project button

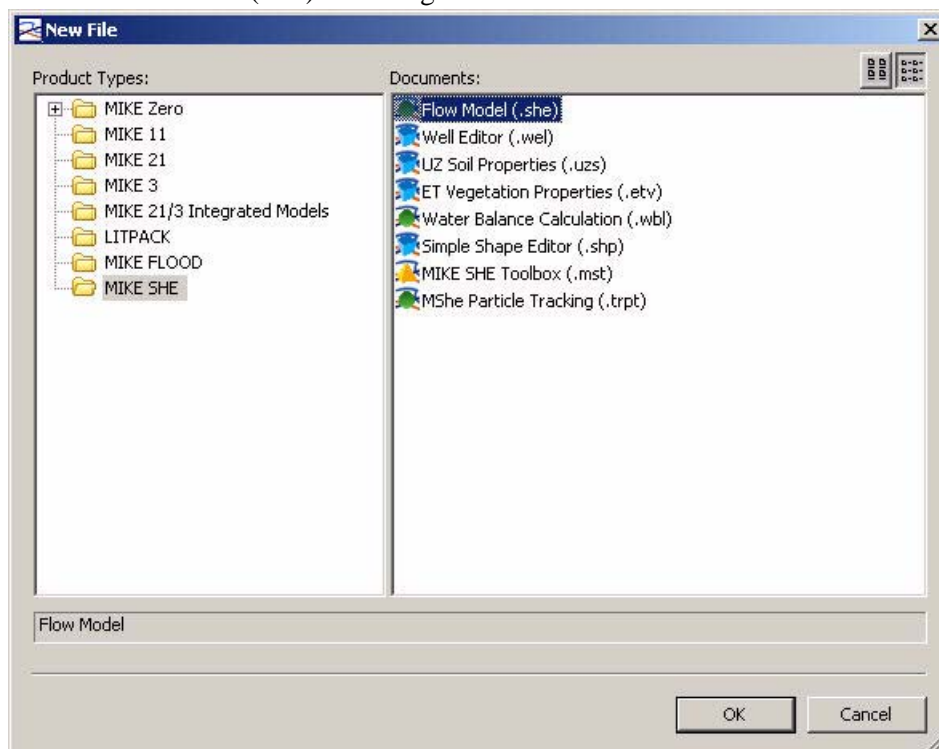
After you are finished with a project, you can use the Delete button to either delete just the .mzp project catalogue file, or the entire project with all of its artifacts. This is a particularly useful function when your project is complete and you have safely archived all of your project data.

### 2.3.2 Creating a New Model File

After starting MIKE Zero, you can create a new MIKE SHE model file by selecting File|New|File... in the top pull-down menu or by using the New File icon, , in the toolbar. When the New File dialogue appears, select MIKE SHE on the left hand side.



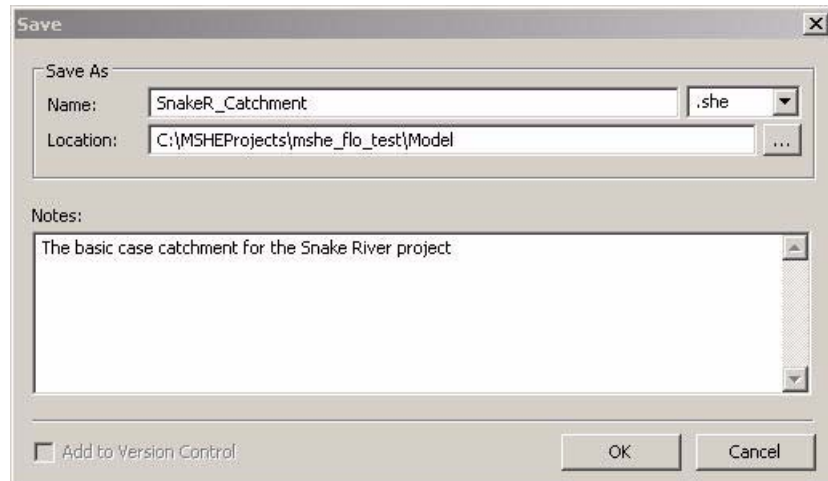
If you want to create new flow model set up file, then select **Flow Model (.she)** on the right hand side and click OK.



Alternatively, you can select any of the listed file types on the right hand side, or even navigate to one of the other items on the left hand side. For example, to create a new dfs0 time series file, you will need to select the Time Series (.dfs0) item from under the MIKE Zero list.



The new MIKE SHE file is created and displayed, but not named until you save the file. So, the first thing you should do is save the file. When you save the file, you will be presented with the following




The default location is the current position in the Project Explorer.

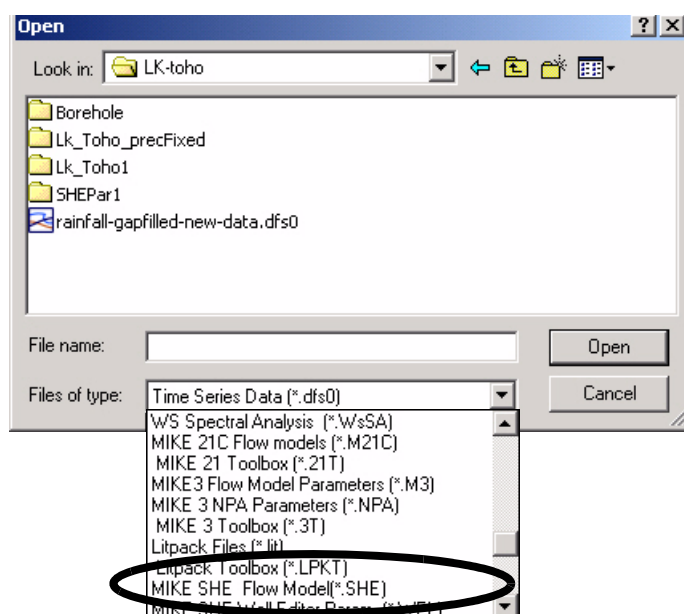
Once the file is saved, you can begin building your model. For more information on building your model, see *Building a MIKE SHE Model (V.1 p. 37)*.

### 2.3.3 Opening an Existing Model File

After MIKE Zero is open, you can open an existing flow model document by

- selecting File|Open in the pull-down menu, or
- clicking on the Open File icon, , in the toolbar, or
- doubling clicking on the file in the Project Explorer.

In the first two cases, the Open File dialogue will appear.



In the File Type combobox is a drop down list of all of the file types that are available. The length of the list depends on which MIKE Zero products you have installed.

Scroll down to and select the file type that you want. This will display a list of available files of that type in the current directory from which you can select the file that you want to open. If your file is in a different directory, you can browse to the correct directory, using the navigation tools at the top. Finally, click Open to open the file.

## 2.4 Importing Existing Release 2005 Projects

You are not required to use the new project functionality in the 2007 Release. If you can simply open any MIKE Zero file in the main MIKE Zero dialogue.

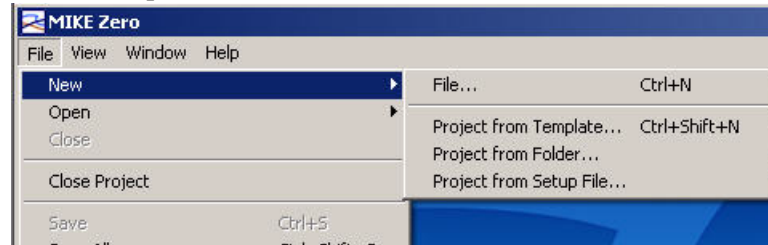
However, to take advantage of the new project focus of the 2007 Release, you will need to import your existing MIKE SHE projects. A MIKE Zero project can be created from an existing 2005 Release MIKE SHE models by either

Reading the directory structure and adding all of the files in the directory and all of the sub-directories, or



Reading the .she file (or any valid MIKE Zero pfs-type file) and adding all of the files referenced in the file.

These two options are accessed from the File/New... menu, where



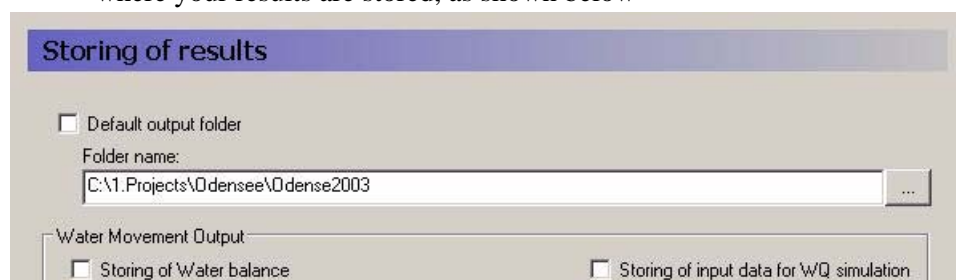
where

- Project from Folder will recurs through the sub-folder structure and add all of the files to the new project, and
- Project from Setup File will add all of the referenced files.

#### 2.4.1 Linking 2005 Results in a 2007 .she Set Up File

The MIKE Zero project template places all Results from your project in a common Results folder. This is different from the 2005 Release, where all of you MIKE SHE results were placed in a folder under your project with the same name as the project. This means that when you open your 2005 project in MIKE SHE you will not be able to access your results in the Results Tab.


To link the Results Tab to your existing results, without rerunning your model, you need to uncheck the Default Output folder option in the Storing of results dialogue. Then use the browse button to specify the directory where your results are stored, as shown below

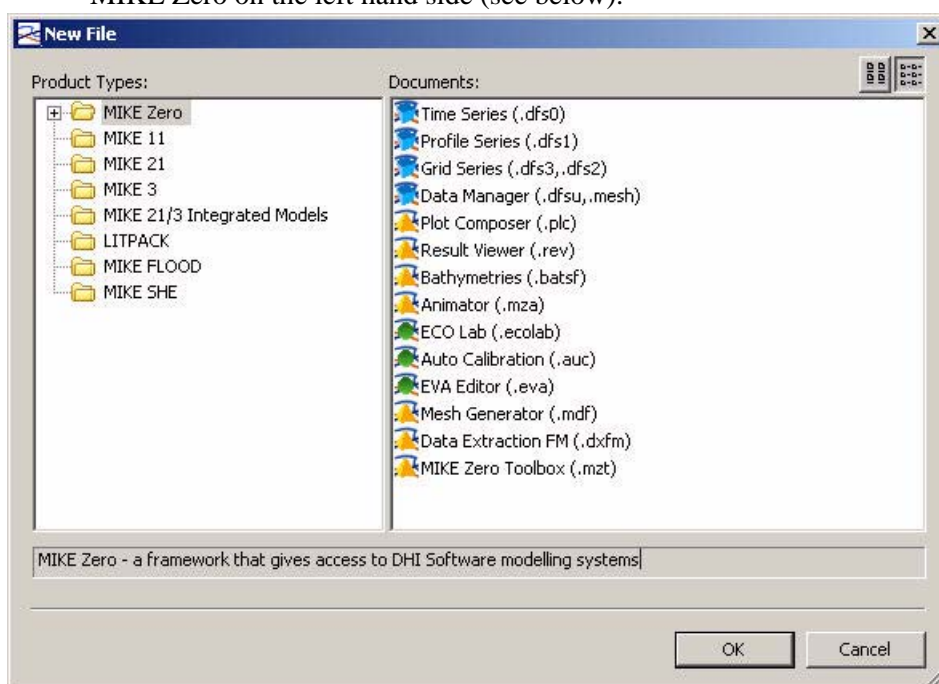




## 2.5 MIKE Zero Editors

The MIKE Zero interface also includes general tools for data editing, analysis and manipulation.

After starting MIKE Zero, you can create a new MIKE Zero file by selecting File|New|File... in the top pull-down menu or by using the New File icon, , in the toolbar. When the New File dialogue appears, select MIKE Zero on the left hand side (see below).



The most commonly used MIKE Zero tools in MIKE SHE are described in the printed manual. However, those tools that are either infrequently used, or irrelevant to MIKE SHE are not included here.

The MIKE Zero tools include (with the tools relevant for MIKE SHE in bold):

- **The Time Series Editor** - for time series data
- The Profile Series Editor - for time varying 1D data (not used in MIKE SHE)
- **The Grid Editor** - for time varying 2D and 3D data
- Data Manager - for finite element data
- **The Plot Composer** - for creating standard report plots





- **The Results viewer (V.1 p. 93)** - for results presentation
- Bathymetries - for sea bed elevations
- Animator - for 3D visualization of surface water and waves
- EcoLAB - for water quality in surface water, which can be used in MIKE 11, but not yet in MIKE SHE
- **Auto Calibration Tool (V.2 p. 359)** - for autocalibration, sensitivity analysis and scenario management
- EVA Editor - for extreme value analysis of surface water flows
- Mesh Generator - for the finite element versions of MIKE 21 and MIKE 3
- Data Extraction FM - for the finite element data
- **MIKE Zero Toolbox** - various tools for data manipulation

## **2.6    *The MIKE SHE User Interface***

The MIKE SHE user interface is organized by task. In every model application you must

- 1 Set up the model,
- 2 Run the model, and
- 3 Assess the results.

The above three tasks are repeated until you obtain the results that you want from the model.



When you create or open a MIKE SHE model, you will find your self in the Setup Tab of the MIKE SHE user interface.

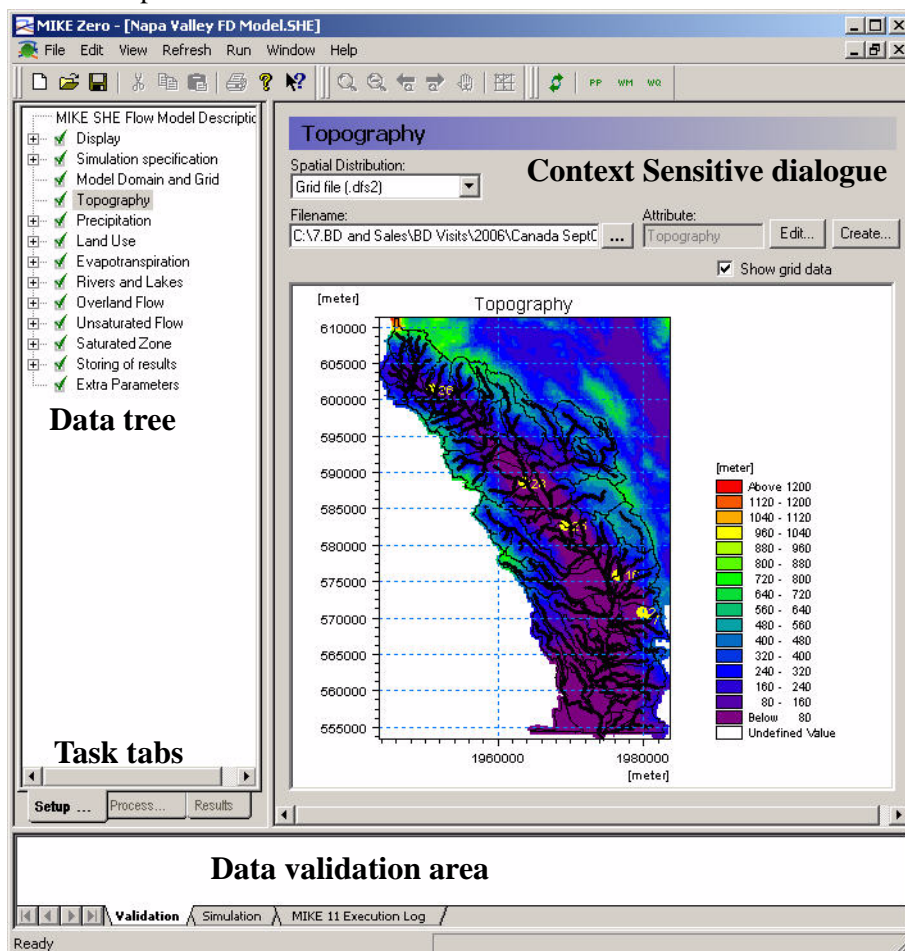


Figure 2.1 Graphical overview of the in the MIKE SHE GUI, without the Project Explorer.

The Setup editor is divided into three sections - the data tree, a context sensitive dialogue and a validation area.

The data tree is dynamic and changes with how you set up your model. It provides an overview of all of the relevant data in your model. The data tree is organized vertically, in the sense that if you work your way down the tree, by the time you come to the bottom you are ready to run your model.

The context sensitive dialogue on the right allows you to input the required data associated with your current location in the data tree. The



dialogues vary with the type of data, which can be any combination of static and dynamic data, as well as spatial and non-spatial data. In the case of spatial and time varying data, the actual data is not input to the GUI. Rather, a file name must be specified and the link to the file is stored in the GUI. Furthermore, the distribution of the data in time and space need not correspond between the various entries. For example, rainfall data may be entered as hourly values and pumping rates as weekly values, while the model may be run with daily timesteps.

The validation area at the bottom of the dialogue provides you with immediate feedback on the validity of the data that you have input.

After you have set up your model, you must switch to the Processed Data tab and run the pre-processing engine on the model. This step reconciles all of the various spatial and time series data and creates the actual data set that will be run by MIKE SHE. Once the data has been pre-processed the simulation can be started. Using the Pre-processing tab at the bottom, you can load and view the pre-processed data.

After the simulation is finished, you can switch to the Results tab, where you can view the detailed time series output as in a report-ready HTML view. Alternatively, you can use the Results Viewer, which is one of the generic MIKE Zero tools, for more customized and detailed analysis of the gridded output.

### **2.6.1 The Setup Data Tree**

Your MIKE SHE model is organized around the Setup Data Tree. The layout of the tree depends on the model components that are active in the current model, which are selected in the Simulation Specification dialogue. Opposite the data tree is the corresponding dialogue for the currently selected tree branch.

The data tree is designed to hide the components that are not needed for the current simulation. However, no data is ever lost if the branch is hidden. That is, all data is retained, even if the branch is not currently visible.

The design of the data tree is such that when you make selections in the current dialogue, the tree is automatically updated to reflect the selection. However, the layout of the data tree and the options available in the current dialogue are such that the data tree will only change along the current branch. That is, if you make a selection in the current dialogue, additional options or branches may become available further along the branch. However, no changes will occur in other branches of the data tree. For example, if you make a selection in the Precipitation dialogue, this will affect



the Precipitation data branch. It will not affect the Evapotranspiration branch.

The only exception to the above rule is selections made in the Simulation Specification dialogue, which is used to set up the entire data tree. Thus, for example, if you unselect Evapotranspiration in the Simulation Specification dialogue, the entire Evapotranspiration branch will disappear.



### 3 ***BUILDING A MIKE SHE MODEL***

The MIKE SHE user interface is organized around the workflow that you must go through to build a model. This chapter is essentially organized around these three main tasks with detail added for the individual tasks.

- 1 Set up the model,
- 2 Run the model, and
- 3 Assess the results.

#### 3.1 ***Adding Background Maps***

Arguably, the first step in building your model is to define where you are going to place your model. This generally involves defining a basic background map for your model area.

The Display item is located at the top of the data tree to make it easy to add and edit your background maps. In the Display item, you can add any number of images to your model setup, in a variety of formats. The images are carried over to the various editors, so you can keep a consistent display between the set up editor and, for example, the Grid Editor and the Results Viewer.

There are a few exceptions to the above functionality.

- The Mike 11 network is not carried over in the Results Viewer, and
- The view settings for the overlays are not carried over into the Results Viewer. This means that all of your overlays, including turned off overlays are visible in the Grid Editor.

In the event that you are using scanned paper maps, if you maps are not rectilinear, or are not correctly georeferenced, then you can use the Image Rectifier (see on-line help under MIKE Zero) to align your image to the coordinate system you are using.

#### 3.2 ***Selecting the Processes***

MIKE SHE allows you to simulate all of the processes in the land phase of the hydrologic cycle. That is, all of the process involving water movement after the precipitation leaves the sky. Precipitation falls as rain or snow depending on air temperature - snow accumulates until the temperature



increases to the melting point, whereas rain immediately enters the dynamic hydrologic cycle. Initially, rainfall is either intercepted by leaves (canopy storage) or falls through to the ground surface. Once at the ground surface, the water can now either evaporate, infiltrate or runoff as overland flow. If it evaporates, the water leaves the system. However, if it infiltrates then it will enter the unsaturated zone, where it will be either extracted by the plant roots and transpired, added to the unsaturated storage, or flow downwards to the water table. If the upper layer of the unsaturated zone is saturated, then additional water cannot infiltrate and overland flow will be formed. This overland flow will follow the topography downhill until it reaches an area where it can infiltrate or until it reaches a stream where it will join the other surface water. Groundwater will also add to the base-flow in the streams, or the flow in the stream can infiltrate back into the groundwater.

The Technical Reference contains detailed information on the numerical methods that can be selected from this dialogue:

- Overland Flow - Reference (V.2 p. 211)
- Channel Flow - Reference (V.2 p. 227)
- Evapotranspiration - Reference (V.2 p. 243)
- Unsaturated Flow - Reference (V.2 p. 261)
- Saturated Flow - Reference (V.2 p. 289)

### **3.3    *Setting up the Simulation parameters***

Once you have selected your processes, then there are several simulation parameters that need to be defined. Although, none of these are initially very important. You can come back to all of these at any time.

However, we recommend that you do set up your simulation period when you first create your model. The simulation period is used to verify all of your time series data to make sure that your time series cover your simulation period. You can still add your time series files, but if your simulation period is not correct, then you will get a warning message in the message field at the bottom of the page and the time series graphs will not display the proper portion of the time series.

In MIKE SHE, all of the simulation input and output is in terms of real dates, which makes it easy to coordinate the input data (e.g. pumping rates), the simulation results (e.g. calculated heads) and field observations (e.g. measured water levels).



### **Solver parameters**

The default solver parameters for each of the processes are normally reasonable and there is usually no reason to change these unless you have a problem with convergence or if the simulation is taking too long to run. For more information on the solver parameters, you should see the individual help sections for the different solvers:

- OL Computational Control Parameters (*V.2 p. 34*)
- UZ Computational Control Parameters (*V.2 p. 39*)
- SZ Computational Control Parameters (*V.2 p. 40*)

### **Time step control**

Likewise, the time step control is important, but the default values are usually reasonable to get your model up and running. Then, you should go back to the Time Step Control (*V.2 p. 30*) dialogue to optimize your simulation time stepping. For more information on time step control, you can go to the help section for the Time Step Control (*V.2 p. 30*) dialogue, or see the Controlling the Time Steps (*V.1 p. 70*) section.

#### **3.3.1 Hot Starting from a previous simulation**

Your MIKE SHE simulation can be started from a hot start file. A hot start file is useful for simulations requiring a long warm up period or for generating initial conditions for scenario analysis.

To start a model from a previous model run, you must first save the hot start data, in the Storing of Results (*V.2 p. 135*) dialogue. In this dialogue, you specify the storing interval for hot start data. Then in the Simulation Period (*V.2 p. 28*) dialogue, you can specify the hot start file and then select from the available stored hot start times.

### **3.4 Defining the model domain and grid**

Regardless of the components included in your model, the first step in your model development is to define the model area. On a catchment scale, the model boundary is typically a topographic divide, a groundwater divide or some combination of the two. In general, there are no constraints on the definition of the model boundaries. However, the model boundaries should be chosen carefully, keeping in mind the boundary conditions that will be used for both the surface water and groundwater components.

All other spatial data defined in the data tree, such as topography, is interpolated during pre-processing to the Model Domain and Grid.



You can define your model domain and the grid using either a DHI grid file (dfs2 format) or a GIS shape file (.shp format).

### Using a dfs2 file

If you define your model domain using a dfs2 grid file, then you must define the cell values as follows:

- Grid cells outside of the model domain must be assigned a delete value - usually  $-1e-35$ .
- Grid cells inside the model domain must be assigned a value of 1.
- Grid cells on the model boundary must be assigned a value of 2.

This distinction between interior grid cells and boundary cells is to facilitate the definition of boundary conditions. For example, drainage flow can be routed to external boundaries but not to internal boundaries.

Since the model domain is defined as part of the dfs2 file format, if you want to change the extent of your model domain, you must edit the .dfs2 file. However, if you want to change the grid spacing, then it is probably easier to create a new file.

**Note** The Model Domain and Grid does not have to have the same dimensions (size and spacing) as other specified dfs2 files. But, the dfs2 files for integer grid codes must be coincident. That is the rows and columns must align with one another. For more information on this see Integer Grid Codes (*V.1 p. 290*).

### Using a shp file

It is much easier to define your Model Domain and Grid via a GIS polygon shape (.shp) file. In this case, the definition of integer code values is taken care of internally. Once you have defined the polygon file to use, then you specify the spatial extent and origin location of the model domain and grid.

An important advantage of using a polygon for the model domain, is that the number of rows and columns can be easily adjusted.

### Creating dfs2 or shp files

There is a Create button next to the Browse button that opens a dialogue where you can define a dfs2 grid file. This utility automatically creates the grid file with the appropriate Item Type.





In this dialogue, you can specify the overall dfs2 grid dimensions and origin. After you have created the file, then you can open and edit the file in the Grid Editor using the Edit button.

### 3.4.1 **MIKE SHE model limits**

Although, there are no theoretical limits to the size of your model, there are practical limits. MIKE SHE does not yet support dynamic allocation of memory, so at the moment there are fixed limits on the size of certain arrays in the MIKE SHE code. The current default array size limits of MIKE SHE are:

- number of cells in x- and y-direction: 700
- number of computational cells per layer (incl. boundary cells): 125,000
- number of computational saturated zone layers: 50
- number of river links: 10,000
- number of computational UZ columns (multi-layer UZ): 30,000
- number of nodes per UZ column (multi-layer UZ): 150

If you bump up against these limits, we can, of course, provide you with a specially compiled version with higher limits. However, if you exceed these limits, you may quickly reach the practical limits of your computer resources. If you think you need a larger version, may we suggest the following:

- First, evaluate your model to see if you really need such a large model, as there are significant costs, in terms of run time, memory usage etc. that are a function of the model size.
- If you still feel that you must have a larger model than the limits allow, then we suggest that you first do a rough calibration with a model within the original limits. This way you can verify that your computer is at least capable of running the current maximum size model. Further, the model independent structure of MIKE SHE makes it reasonable to refine your model later with a minimum of effort. This is probably a good idea, in any case, since the full model may require significantly more resources and it will always be easier to roughly calibrate a smaller model and use the rough calibration as an initial condition for the full model.



- Finally, when you want to run the larger model, tell us which array you need changed and by how much. Also send us the number of rows and columns in your model, as well as a list of the numerical engines being used. We will send you a new compiled version to meet your specifications.

### 3.4.2 **MIKE SHE Demo model limits**

If no dongle is installed, or if a current license file is not available, then MIKE SHE will run in demo model. In this case, the model size limits are more restrictive. If you need a full size MIKE SHE to perform your evaluation, then you are welcome to contact your local DHI office to request a 30-day evaluation licenses.

The current demo restrictions are as follows:

- number of cells in x- and y-direction: 40
- number of computational cells per layer (incl. boundary cells): 550
- number of computational saturated zone layers: 2
- number of river links: 150
- number of computational UZ columns (multi-layer UZ): 155
- number of nodes per UZ column (multi-layer UZ): 100
- simulation time: 4444 hours or 185 days
- number of UZ timesteps: 800
- number of SZ timesteps: 200
- no steady-state SZ
- no irrigation

## 3.5 **Defining Topography**

In MIKE SHE, the topography defines the upper boundary of the model. The topography is used as the top elevation of both the UZ model and the SZ model. The topography defines the drainage surface for overland flow.

Many of the elevation parameters can be defined relative to the topography by means of a checkbox in the dialogue, including

- Lower Level (Geological Layer or Lense, or Water Quality Layer) (*V.2 p. III*),



- Upper Level (*V.2 p. 111*),
- Lower Level (Numerical Layer) (*V.2 p. 117*),
- Initial Potential Head (*V.2 p. 117*), and
- Drain Level (*V.2 p. 125*).

Depth parameters, such as ET Surface Depth (*V.2 p. 100*), are also measured from the topography.

Topography is typically defined from a DEM, defined from either a point theme shape file, or an ASCII file. If you have an ArcGIS Grid DEM, this can be converted to a dfs2 file using the MIKE Zero Toolbox. Surfer Grid files can be saved as an ASCII xyz files and then interpolated in MIKE SHE.

Non-dfs2 files or dfs2 files that have a different grid definition than the model grid are all interpolated to the grid defined in the Model Domain and Grid.

The Bilinear Interpolation (*V.1 p. 292*) method is useful for interpolating previously gridded DEM data. Whereas, the Triangular Interpolation (*V.1 p. 296*) method is useful for contour data digitized from a DEM.

### 3.6 Adding Precipitation

The precipitation rate is the measured rainfall. If Snowmelt (*V.2 p. 57*) is included and the Air Temperature (*V.2 p. 60*) is below the Threshold melting temperature (*V.2 p. 62*) then the precipitation will accumulate as snow.

You can specify the precipitation as a rate, for example in [mm/hr], or as an amount, for example in [mm]. If you use the amount method, MIKE SHE will automatically convert this to a rate during the simulation.

If you use a rate, then the EUM Data Units (*V.1 p. 349*) must be Precipitation and the time series must be Mean Step Accumulated (*V.1 p. 245*).

If you use an amount, then the EUM Data Units must be Rainfall and the time series must be Step Accumulated (*V.1 p. 245*).

The Precipitation Rate item comprises both a distribution and a value. The distribution can be either uniform, station-based or fully distributed. If the data is station-based then for each station a sub-item will appear where you can enter the time series of values for the station.



### 3.7 Adding Surface Water

The Rivers and Lakes dialogue allows you to specify the MIKE 11 .sim11 file that you want to use in your model.

Detailed information on how to couple MIKE 11 and MIKE SHE and the overbank spilling options can be found in the chapter

- Coupling MIKE 11 and MIKE SHE (*V.1 p. 163*),

whereas detailed information on the coupling mechanism can be found in

- Channel Flow - Reference (*V.2 p. 227*).

### 3.8 Adding Overland Flow

Overland flow is required when you are using MIKE 11 in MIKE SHE, as the overland flow module provides lateral runoff to the rivers. If you don't want to simulate overland flow, then you can specify a Mannings M of 0, which will disable overland flow.

The overland flow can be calculated using either a semi-distributed method or a finite difference method using the diffusive wave approximation. The finite difference method should be used when you are interested in calculating detailed overland flow, while the semi-distributed, simplified method should be used for regional applications, where detailed overland flow is not required.

More information on the numerical methods can be found in

- Finite Difference Method (*V.2 p. 211*)
- Diffusive Wave Approximation (*V.2 p. 211*)
  - Successive Over-Relaxation (SOR) Numerical Solution (*V.2 p. 216*)
  - Explicit Numerical Solution (*V.2 p. 217*)
- Simplified Overland Flow Routing (*V.2 p. 220*)

Whereas, detailed information on the coupling between MIKE 11 and MIKE SHE and the overbank spilling options can be found in the chapter

- Coupling MIKE 11 and MIKE SHE (*V.1 p. 163*)



### 3.8.1 Overland Flow Boundary Conditions

The outer boundary condition for the overland flow solver is a specified head, based on the initial water depth in the outer nodes of the model domain. Thus, if the water depth inside the model domain is greater than the initial depth on the boundary, water will flow out of the model. If the water depth is less than the initial depth on the boundary, the boundary will act as a source of water.

No flow boundaries on the outer model domain can be implemented using the Separated Flow Areas option in the Setup Editor for the Overland Flow module. The Separated Flow areas are used to prevent overland flow from flowing between cells that are separated by topographic features, such as dikes, that cannot be resolved within a the grid cell. If you define the separated flow areas along the intersection of the inner and outer boundary areas, MIKE SHE will keep all overland flow inside of the model.

## 3.9 Adding Unsaturated Flow

### 3.9.1 UZ Classification

Since UZ computations in all grid squares for most large-scale applications requires excessive computation time, MIKE SHE enables you to compute the UZ flow in a reduced subset of grid squares. The subset classification is done automatically by the pre-processing program according to soil types, vegetation types, climatic zones, and depth to the groundwater table.

- **Automatic classification** The automatic classification requires a distribution of groundwater elevations (see Groundwater Depths used for UZ Classification). This can be either the initial depth to the groundwater based on the initial heads, or you can supply a .dfs2 map of the groundwater elevations. In both cases, you must supply a table of intervals upon which the classification will be based. The number of computational columns depends on how narrow the intervals are specified. If, for example, two depths are specified, say 1 m and 2 m, then the classification with respect to the depth to groundwater will be based on three intervals: Groundwater between 0 m and 1 m, between 1 m and 2 m, and deeper than 2 m.

One tip is to extract a map of the calculated potential head in the very upper saturated zone layer from a previous simulation. The map should represent the time of the year when the largest variations of the groundwater table are expected (deep groundwater in the hills and shallow



groundwater close to the rivers). Repeat the procedure as calibration improves.

If the Linear Reservoir method is used for the groundwater, then the Interflow reservoirs are also used in the classification. However, since feedback to the UZ only occurs in the lowest Interflow reservoir of each subcatchment, the Interflow reservoirs are added to the Automatic Classification in two zones - those that receive feedback and those that don't.

- **Specified classification** Alternatively a data file specifying Integer Grid Codes, where UZ computations are carried out can be specified, with grid codes range from 2 up to the number of UZ columns (see Specified classification). The location of the computational column is specified by a negative code and the simulation results are then transferred to all grids with the an equivalent positive code. For example, if a grid code holds the value -2 a UZ computation will be carried out for the profile located in that grid. Simulation results will subsequently be transferred to all grid codes with code value 2. An easy way to generate a .dfs2 file to be used for specification of UZ computational columns is to let the MIKE SHE setup program generate an automatic classification first, and subsequently extract the UZ classification grid codes. The extracted .dfs2 file can be edited in the 2D editor as desired and used to specify UZ computational grids.
- **Calculated in all Grid points** For smaller scale studies, or studies where the classification system becomes intractable, you can specify that computations are to be carried out in all soil columns.
- **Partial Automatic** Finally a combination of the Automatic classification and the Specified classification is available. If this option is chosen an Integer Grid Code file must be provide (see Partial automatic classification) with the following grid codes: In grid points where automatic classification should be used the grid code 1 must be given. In grid points where computation should be performed for all cells the grid code 2 must be given.

### **3.9.2 Coupling Between Unsaturated and Saturated Zone**

The following procedure should be used to ensure that the unsaturated zone does not drop below the bottom of the first calculation layer of the saturated zone:

- After a simulation, create a map of grid statistics of the potential head in the first calculation layer of the saturated zone



- Subtract the map of the minimum potential head from the map of the bottom level of the first calculation layer of the saturated zone.
- View the difference map. If the difference is very small in some areas of the map (e.g.  $<0.5$  m), it is strongly advised to move the bottom level of the first calculation layer of the saturated zone downwards.
- Repeat this procedure until there are no small differences.

The water balance program can be used to get an overview of errors due to a bad setup of the unsaturated zone. The follow procedure can be used to make a map of UZ-errors:

- Create a sub catchment map by retrieving UZ-classification codes from the input file.
- Replace negative values of the classification code map by positive values in the 2D graphical editor.
- Use the sub catchment map in the water balance setup file to make a UZ map of the water balance, which will create your map of UZ-errors.

**Vertical discretisation** - The vertical discretisation of the soil profile typically contains small cells near the ground surface and increasing cell thickness with depth. However, the soil properties are averaged if the cell boundaries and the soil boundaries do not align. .

The discretisation should be tailored to the profile description and the required accuracy of the simulation. If the full Richards equation is used the vertical discretisation may vary from 1-5 cm in the uppermost grid points to 10-50 cm in the bottom of the profile. For the Gravity Flow module, a coarser discretisation may be used. For example, 10-25 cm in the upper part of the soil profile and up to 50-100 cm in the lower part of the profile. Note that at the boundary between two blocks with different cell heights, the two adjacent boundary cells are adjusted to give a smoother change in cell heights.

## 3.10 Adding Groundwater

### 3.10.1 Your Conceptual Geologic Model

The development of the geological model is probably the most time consuming part of the initial model development. Before starting this task, you should have developed a conceptual model of your system and have at your disposal digital maps of all of the important hydrologic parameters, such as layer elevations and hydraulic conductivities.



In MIKE SHE you can specify your subsurface geologic model independent of the numerical model. The parameters for the numerical grid are interpolated from the grid independent values during the preprocessing.

The geologic model can include both geologic layers and geologic lenses. The former cover the entire model domain and the latter may exist in only parts of your model area. Both geologic layers and lenses are assigned geologic parameters as either distributed values or as constant values.

The alternative is to define the hydrogeology based on geologic units. In this case, you define the distribution of the geologic units and the geologic properties are assigned to the unit.

Each geologic layer can be specified using a dfs2 file, a .shp file or a distribution of point values. However, you should be aware of the way these different types of files are interpolated to the numerical grid.

The simplest case is that of distributed point values. In this case, the point values are simply interpolated to the numerical grid cells based on the available interpolation methods.

In the case of shp files, at present, only point and line theme .shp files are supported. Since lines are simply a set of connected points, the .shp file is essentially identical to the case of distributed point values. Thus, it is interpolated in exactly the same manner.

The case of .dfs2 files is in fact two separate cases. If the .dfs2 file is aligned with the model grid then the cell value that is assigned is calculated using the bilinear method with the 4 nearest points to the centre of the cell. If the .dfs2 file is not aligned with the model grid then the file is treated exactly the same as if it were a .shp file or a set of distributed point values.

The geologic model is interpolated to the model grid during preprocessing, by a 2 step process.

- 1 **The horizontal geologic distribution is interpolated to the horizontal model grid.** If Geologic Units are specified then the integer grid codes are used to interpret the geologic distribution of the model grid. If distributed parameters are specified then the individual parameters are interpolated to the horizontal model grid as outlined above.
- 2 **The vertical geologic distribution is interpolated to the vertical model grid.** In each horizontal model grid cell, the vertical geologic model is scanned downwards and the soil properties are assigned to the cell based on the average of the values found in the cell weighted by





the thickness of each of the zones present. Thus, for example, if there were 3 different geologic layers in a model cell each with a different Specific Yield, then the Specific Yield of the model cell would be

$$S_y = \frac{S_{y1} \cdot z_1 + S_{y2} \cdot z_2 + S_{y3} \cdot z_3}{z_1 + z_2 + z_3} \quad (3.1)$$

where  $z$  is the thickness of the geologic layer within the numerical cell.

### Conductivity values

Hydraulic conductivity is a special parameter because it can vary by many orders of magnitude over a space of only a few meters or even centimeters. This necessitates some special interpolation strategies.

**Horizontal Interpolation** - The horizontal interpolation of hydraulic conductivity interpolates the raw data values. Thus, in Step 1 above, when interpolating point values that range over several orders of magnitude, such as hydraulic conductivity, the interpolation methods will strongly weight the larger values. That is, small values will be completely overshadowed by the large values.

In fact, the interpolation in this case should be done on the logarithm of the value and then the cell values recalculated. Until this option is available in the user interface, you should interpolate conductivities outside of MIKE SHE using, for example, Surfer. Alternatively, the point values could be input as logarithmic values and the Grid Calculator Tool in the MIKE SHE Toolbox can be used to convert the logarithmic values in the .dfs2 file to conductivity values.

**Vertical Interpolation** - In Step 2 above, the geologic model is scanned down and interpreted to the model cell. Although, horizontal conductivity can vary by several orders of magnitude in the different geologic layers that are found in a model cell, the water will flow horizontally based on the highest transmissivity. Thus, the averaging of horizontal conductivity can be done the same as in the example for Specific Yield above. Vertical flow, however, depends mostly on the lowest hydraulic conductivity in the geologic layers present in the model cell. In this case a harmonic weighted mean is used instead. For a 3 layer geologic



model in one model cell, the vertical conductivity would be calculated by

$$K_z = \frac{z_1 + z_2 + z_3}{\frac{z_1}{K_{z1}} + \frac{z_2}{K_{z2}} + \frac{z_3}{K_{z3}}} \quad (3.2)$$

where  $z$  is the thickness of the geologic layer within the numerical cell.

### 3.10.2 Working with Lenses

In building a geologic model, it is typical to find discontinuous layers and lenses within the geologic units. The MIKE SHE setup editor allows you to specify such units - again independent of the numerical model grid.

Lenses are specified by defining either a .dfs grid file or a polygon .shp file for the extents of the lenses. The .shp file can contain any number of polygons, but the user interface does not use the polygon names to distinguish the polygons. If you need to specify several lenses, you can use a single file with many polygons and specify distributed property values, or you can specify multiple individual polygon files, each with unique property values.

In the case of lenses, an extra step is added to the beginning of the 2-step process outlined in the previous section. The location of the lenses is first interpolated to the horizontal numerical grid. Then the lenses become essentially extra geologic layers in the columns that contain lenses. However, there are a number of special considerations when working with lenses in the geologic model.

- **Lenses override layers** - That is, if a lense has been specified then the lense properties take precedence over the layer properties and a new geologic layer is added in the vertical column.
- **Vertically overlapping lenses share the overlap** - If the bottom of lense is below the top of the lense beneath, then the lenses are assumed to meet in the middle of the overlapping area.
- **Small lenses override larger lenses** - If a small lense is completely contained within a larger lense the smaller lense dominates in the location where the small lense is present.
- **Negative or zero thicknesses are ignored** - If the bottom of the lense intersects the top of the lense, the thickness is zero or negative and the lense is assumed not to exist in this area.



### 3.10.3 Numerical Layers

#### Boundary Conditions

The upper boundary of the top layer is always either the infiltration/exfiltration boundary, which in MIKE SHE is calculated by the unsaturated zone component or a specified fraction of the precipitation if the unsaturated zone component is excluded from the simulation.

The lower boundary of the bottom layer is always considered as impermeable.

In MIKE SHE, the rest of the boundary conditions can be divided into two types: Internal and Outer. If the boundary is an outer boundary then it is defined on the boundary of the model domain. Internal boundaries, on the other hand, must be inside the model domain.

### 3.10.4 Groundwater Drainage

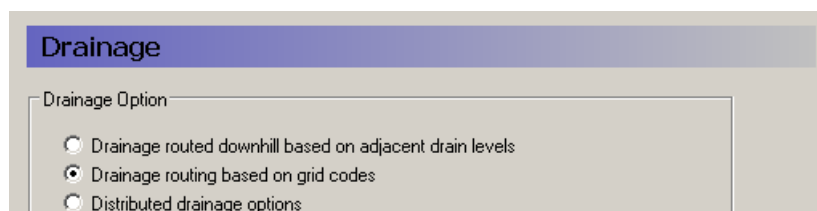
Surface drainage is a special boundary condition in MIKE SHE used to defined natural and artificial drainage systems that cannot be defined in the MIKE 11 River setup. It can also be used to simulate overland flow in a simple lumped conceptual approach. Surface drainage is applied to the layer of the Saturated Zone model containing the drain level. Water that is removed from the saturated zone by surface drainage is routed to local surface water bodies.

Drainage flow is simulated using an empirical formula, which requires, for each cell, a drainage level and a time constant (leakage factor) that are used for routing the water out of the element. Both drain levels and time constants can be spatially defined. A typical drainage level is 1m below the ground surface and a typical time constant is between  $1e-6$  and  $1e-7$  1/s.

MIKE SHE also requires a reference system for linking the drainage to a recipient cell, which can be a MIKE 11 river node, another SZ grid cell, or a model boundary. Whenever drain flow is produced during a simulation, the computed drain flow is routed to the recipient point using a linear reservoir routing technique.

#### Drainage references

There are four different options for setting up the reference system for drainage



### Drainage routed downhill based on adjacent drain levels

This option was originally the only option in MIKE SHE. The reference system is created automatically using the slope of the drains calculated from the drainage levels in each cell. Thus, as long as a downward slope is found, the drain flow will continue until crossing a river or the model boundary.

If local depressions in the drainage levels exist, the SZ nodes in these depressions may become the recipients for a number of drain flow producing nodes. This often results in the creation of a lake at such local depressions.

This drain-slope based reference system has been used in MIKE SHE for many years and works well in most situations. However, when MIKE SHE is applied where there is very little surface topographic relief, it is often difficult to establish a suitable reference system based on the surface topography/drain slopes. For example, often it is assumed that the drains are located 0.5 or 1 meter below the terrain. In flat areas, this may generate many undesired local depressions, which may receive drainage water from a large area, thus generating lakes in places where there should not be a lake. MIKE SHE considers a grid point to be a local depression even if the drainage level in the four surrounding model grids is only 1 mm higher. The only way to avoid such problems is to create a drain level map that does not contain “wrong” local depressions. For large models this may be difficult and time consuming. In this case, one of the other drainage options may be better.

**Note:** This method is not allowed when using Time varying drainage parameters (*V.I p. 152*).

### Drainage routing based on grid codes

This method is often used when the topography is very flat, which can result in artificial depressions, or when the drainage system is very well defined, such as in agricultural applications.

In this method, the drainage levels and the time constants are defined as in the previous method. However, a grid code map is also required, which is



used to link the drain flow producing cells to a recipient node. The drain levels are still used to calculate the amount of drain flow produced in each node, but the routing is based only on the code values in the drain code file.

### Distributed drainage options

Choosing this method, adds the Option Distribution item to the data tree. With the Option Distribution, you can specify an integer grid code distribution that can be used to specify different drainage options in different areas of your model. If the grid code equals 1, then the first option is used. If the grid code equals 2, then the second method above is used. If the grid code equals 3, then the drainage can be routed directly to a particular MIKE 11 branch. If the grid code equals 4, then the drainage can be routed to a particular MOUSE manhole.

### Drain flow to boundaries

The fourth option simply exports drainage water out of the model.

## Drain Codes

If the drainage routing is specified by Drain Codes, a grid code map is required that is used to link the drain flow producing cells to recipient grid cells. The drain levels are still used to calculate the amount of drain flow produced in each node, but the routing is based only on the code values in the drain code file.

The Drain Code can be any integer value, but the different values have the following special meanings:

**Code = 0** - Grid cells with an Drain Code value of zero will not produce any drain flow and will not receive any drain flow.

**Code > 0** - Grid cells with **positive** Drain Code values will drain to the nearest river, boundary or local depression in the drain level - in that priority - located next to a cell with the same Drain Code value. Thus, if a grid cell produces drainage,

1. If there are one or more cells with the same drain code next to a river link, then the drain flow will be routed to the nearest of these cells.
2. If there are no cells with the same Drain Code located next to a river link, then the drain flow will be routed to the nearest boundary cell with the same Drain Code value.



3. If there are no boundary cells with the same Drain Code value, the drain flow will be routed to the cell with the lowest drain level that has the same Drain Code value (which may create a lake).

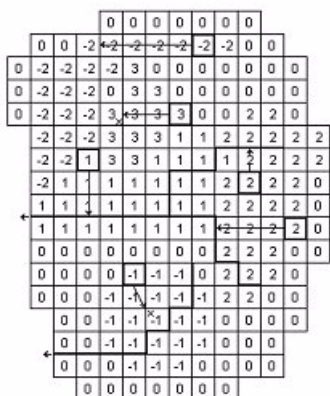
**Code < 0** - Grid cells with **negative** Drain Code values will drain to either a boundary or a local depression, in that order. Thus, if a grid cell produces drainage,

1. If there are no cells with the same Drain Code located next to a river link, then the drain flow will be routed to the nearest boundary cell with the same Drain Code value.

2. If there are no boundary cells with the same Drain Code value, the drain flow will be routed to the cell with the lowest drain level that has the same Drain Code value (which may create a lake).

One method that is often used is to specify only one Drain Code for the entire model area (e.g. Drain Code 1). Thus, all grids can drain and any drain flow is routed to the nearest river link. If there are no rivers, the drain flow will be routed to the nearest boundary. If you want to route all drain flow to the boundaries instead of the rivers, a negative drain code can be specified for the entire area (e.g. Drain Code -1).

### Example



- The grid cells with Drain Code 3 drain to a local depression since no boundary or river link is found adjacent to a grid with the same drain code.
- The grid cells with Drain Code 1 or 2 drain to nearest river link located adjacent to a grid with the same drain code.



- The grid cells with drain code 0 do not contain drains and thus no drainage is produced.
- The grid cells with Drain Code -1 drains to local depression since no boundary is found adjacent to a grid with the same drain code.
- The grid cells with Drain Code -2 drains to nearest boundary grid with the same drain code.

### Option Distribution

The drain type distribution is used to distinguish areas of the model where different drainage options are used.

**Code = 1** - Drainage in grid cells with a value of 1 is routed downhill based on the value of the drain level specified in Drain Level data item.

**Code =2** - Drainage in grid cells with a value of 2 is routed via Drain Codes as specified in the Drain Codes data item.

**Code = 3** - Drainage in grid cells with a value of 3 is routed to a specified MIKE 11 branch and chainage. At the moment, this options requires the use of Extra Parameters.

**Code = 4** - Drainage in grid cells with a value of 4 is routed to a specified MOUSE man hole. At the moment, this options requires the use of Extra Parameters.

### 3.10.5 MIKE SHE versus MODFLOW

The MIKE SHE can be used to simulate all of the processes in the land phase of the hydrologic cycle, including overland flow, channel flow, groundwater flow in the unsaturated zone and saturated groundwater flow. MODFLOW, on the other hand, is restricted to simulating flow only in the saturated groundwater zone. Although many of the processes simulated in MIKE SHE are used in a similar way when simulating groundwater flow with MODFLOW, they are not actually “simulated” by MODFLOW.

Let’s take groundwater recharge as an example. MODFLOW allows you to include recharge as an upper boundary condition to the groundwater model, where recharge is defined as the amount of water reaching the groundwater table after accounting for evapotranspiration, surface runoff and changing storage in the unsaturated zone. In MODFLOW, the modeller has to account for these processes herself - usually by applying a constant rule-of-thumb fraction to the measured precipitation data. In most cases, the model results are very sensitive to this fraction and since the modeller has little data on this fraction, she will assume an initial value and use this parameter as a calibration parameter. Thus, she will adjust the



amount of recharge during the calibration process until the measured groundwater levels match the calculated values.

However, the fraction of precipitation reaching the groundwater table is constant in neither space nor time. The actual amount of precipitation reaching the groundwater table depends strongly on the maximum rate of infiltration, which is a characteristic of the soil and will vary spatially over the model domain. Further, since the maximum rate occurs when the soil is saturated, different amounts of water will infiltrate during wet periods compared to dry periods. To complicate matters further, the length of the preceding dry period will determine the amount of available storage in the unsaturated zone. For example, if there has been a long dry summer period, then evapotranspiration may have created a large deficit of water in the unsaturated zone that must be satisfied before any water reaches the water table.

This example shows that infiltration of precipitation is a very dynamic process. It depends on a complex interaction between precipitation, unsaturated zone soil properties and the current soil moisture content, as well as vegetation properties.

In MIKE SHE, the saturated zone is only one component of an integrated groundwater/surface water model. The saturated zone interacts with all of the other components - overland flow, unsaturated flow, channel flow, and evapotranspiration.

In comparison, MODFLOW only simulates the saturated flow. All of the other components are either ignored (e.g. overland flow) or are simple boundary conditions for the saturated zone (e.g. evapotranspiration).

On the other hand, there are very few difference between the MIKE SHE Saturated Zone numerical engine and MODFLOW. In fact, they share the same PCG solver. The differences that are present are limited to differences in the discretisation and to some differences in the way boundary conditions are defined.

Setting up the saturated zone hydraulic model involves defining the:

- the geological model,
- the vertical numerical discretisation,
- the initial conditions, and
- the boundary conditions.





In the MIKE SHE GUI, the geological model and the vertical discretisation are essentially independent, while the initial conditions are defined as a property of the numerical layer. Similarly, subsurface boundary conditions are defined based on the numerical layers, while surface boundary conditions such as wells, drains and rivers (using MIKE 11) are defined independently of the subsurface numerical layers.

The use of grid independent geology and boundary conditions provides a great deal of flexibility in the development of the saturated zone model. Thus the same geological model and many of the boundary conditions can be re-used for different model discretisation and different model areas.

### 3.10.6 Importing a MODFLOW 96 or MODFLOW 2000 Model

A FORTRAN executable is automatically installed with MIKE SHE and located in the MIKE SHE bin directory. The program can be used to read a MODFLOW file set and extract the stationary distributed data to a set of point theme shape files. The shp files can then be used directly in MIKE SHE.

To extract data from a MODFLOW model, open a command prompt in the directory containing the input files. On the command prompt line, type

```
MShe_ModflowExtraction //apv <pfs_file_name>
```

where the '//apv' tells the program to show a progress window. Otherwise the execution will be silent. The *pfs\_file\_name* variable is the input file for the MODFLOW extractor. The input file has the standard MIKEZero Pfs format. The input fields of the file are explained below. Lines starting with '/' are not read, but rather can be used as comment lines.

**Note** The '|' around the name-file name and the path of the specified file name must be relative to the location of the pfs file. Below is an example .pfs file for the MODFLOW data extractor program:

```
[MIKESHE_ModflowExtraction]
  FileVersion = 1
  ModflowModel = 'MODFLOW-96'
  NameFileName = |.\Airport5.nam|
  XMin = 300.
  YMin = 400.
  XMax = 3032.
  YMax = 1132
  TimeUnit = 'DAYS'
  LengthUnit = 'METER'
EndSect // MIKESHE_ModflowExtraction
```



The `ModflowModel` variable should be changed to `MODFLOW-2000`, if the MODFLOW model is a MODFLOW 2000 model.

The `NameFileName` is the name of the MODFLOW name file that contains all of the references to the other input files. The minimum and maximum (X,Y) coordinates are used to determine the exact spatial coordinates of the nodal points.

The `TimeUnit` and `LengthUnit` variables are not currently used, but must be input. Valid values for `TimeUnit` are DAYS, HOURS, MINUTES and SECONDS. Valid values for `LengthUnit` are METER and FEET.

**Note** MODFLOW does not have any internal unit checking. The units written in the MODFLOW file are only for display purposes. Also, the units that you define in your MODFLOW user interface may not be the same as those written to the MODFLOW files. So, you need to be careful of units and know what units the MODFLOW files are written in.

The MODFLOW name file has the usual MODFLOW format. However, you should

- Specify a new name for the LIST file in order not to overwrite the LIST file of an existing simulation, and
- Make copies of or rename all output files (lines starting with DATA). Existing result files might otherwise be overwritten during the execution.

For a MODFLOW model, the extraction routine reads and outputs the following MODFLOW static parameters:

Top, Bot, Shead, Tran, Hy, Vcont, Sf1, and Sf2

Plus, it outputs the Specific storage, which is calculated as Sf1 divided by the layer thickness.

The extraction routine outputs point theme shape files -one file per data type - with one item for each extracted layer. The shape file names reflect the MODFLOW manual naming convention (Top.shp, Vcont.shp, etc.). The points represent the centre of each grid square. The model orientation is calculated from the user-specified coordinates of lower left (origin) and upper right corner of the model.

To use the MODFLOW data in MIKE SHE, select the Point/Line .shp option for the static variable. Then browse to the appropriate .shp file. The .shp file will contain one item for each model layer in the MODFLOW



model. The appropriate item is selected in the file browse dialogue. Once the file has been assigned, MIKE SHE will automatically interpolate the data to the model grid.

### Internal inactive zones

Currently, it is not possible to extract the inactive zones from the MODFLOW model and convert these to inactive cells in MIKE SHE. MODFLOW and MIKE SHE treat internal inactive zones quite differently. In MIKE SHE, the internal inactive zones are simply treated as cells with a very low hydraulic conductivity, whereas, MODFLOW ignores them in the solution. Furthermore, the extraction program only writes points to the .shp file for the active nodes. Thus, when it comes to the interpolation in MIKE SHE, the interpolation does not know about the inactive zone and interpolates through the inactive zone - there are simply no data points in the inactive zones.

## 3.11 Setting up your results

### 3.11.1 Importing an ASCII file for Detailed Time Series Output

#### Importing data

Detailed MIKE SHE Time Series data can be imported directly into the Detailed MIKE SHE Time Series dialogue using the Import button. The data file must be a tab-delimited ASCII file without a header line. The file must contain the following fields and be in the format specified below.

```
Name>data typeCode>NewPlot>X >Y >Depth>UseObsdata>dfs0Filename>dfs0ItemNumber
```

where the > symbol denotes the Tab character and

**Name** - is the user specified name of the observation point. This is the name that will be used for the time series item in the Dfs0 file created during the simulation.

**data typeCode** - This is a numeric code used to identify the output data type. See the list of available Data Type Codes in Table 6.1 and Table 6.2 under Output Items (*V.1 p. 85*).

**NewPlot** - This is a flag to specify whether a new detailed time series HTML-plot will be created on the Results Tab:

0 = the output will be added to the previous plot.

1 = Create a new plot



**X, Y** - This is the (X, Y) map coordinates of the point in the same EUM units (ft, m, etc.) as specified in the EUM Database for Item geometry 2-dimensional. (see EUM Data Units)

**Depth** - This is the depth of the observation point below land surface for subsurface observation points. The value is in same EUM units (ft, m, etc.) as specified in the EUM Database for Depth Below Ground (see EUM Data Units). A depth value must always be included, even if not needed.

**UseObsData** - This is a flag to specify whether or not an observation file needs to be input : 0 = No; 1 = Yes

**dfs0FileName** - This is the file name of the dfs0 time series file with observation data. The path to the dfs0 file must be relative to the directory containing the MIKE SHE \*.she document. The .dfs0 extension is added to the file name automatically and should be not be included in the file name. For example, the following input line

```
.\Time\Calibration\GroundwaterObs
```

refers to the file GroundwaterObs.dfs0 located in the subdirectory Time\Calibration, which is found in the same directory as the .she model document.

**dfs0ItemNumber** - This is the Item **number** of the observation data in the specified DFS0 file.

### Example

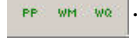
The following is a simple example of a tab delimited ASCII file with two MIKE SHE observation points, where the file containing the observations is called obsdata.dfs0:

|       |    |   |         |         |     |   |                |   |
|-------|----|---|---------|---------|-----|---|----------------|---|
| Obs_1 | 20 | 1 | 234500. | 456740. | 0.  | 0 | .\time\obsdata | 1 |
| Obs_2 | 15 | 1 | 239700. | 458900. | 10. | 1 | .\time\obsdata | 2 |
| Obs_3 | 16 | 0 | 241500. | 459310. | 20. | 1 | .\time\obsdata | 3 |



## 4 ***RUNNING YOUR MIKE SHE MODEL***

In the top icon bar, there is a three-button set of icons for running your model.



**PP** - The PP button starts the preprocessing. You must first **PreProcess** your model data to create the numerical model from your grid independent data. See Preprocessing your model (*V.1 p. 61*).

**WM** - The WM button starts the **Water Movement** simulation. You can only run your water movement simulation after you have preprocessed your data. See Running your Model (*V.1 p. 68*).

**WQ** - The WQ button starts the **Water Quality** simulation. After you have successfully run a water movement simulation to completion, you can run a water quality simulation.

### 4.1 ***Preprocessing your model***

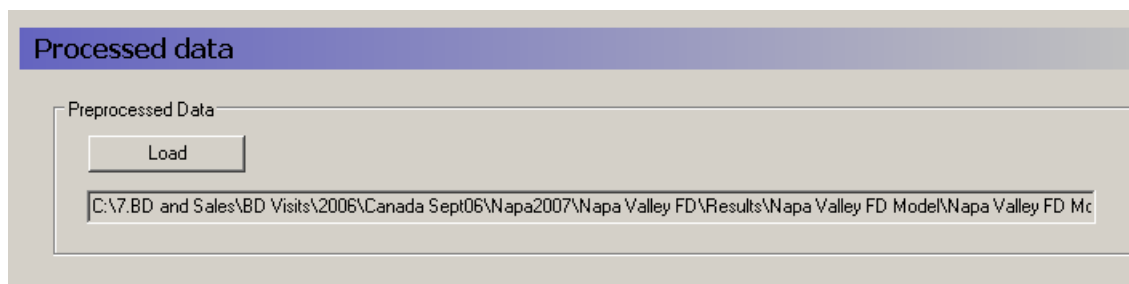
In the Setup Tab, you specify the input data required by the model - including the Model Domain and Grid. However, most of the Setup Data is independent of the Model Domain and Grid. When you pre-process your model set up, MIKE SHE's pre-processor program scans through your model set up and interpolates all spatial data to the specified model domain and grid. This interpolated set up data is stored in a .fif file, which is read during the simulation by the MIKE SHE engine. However, the .fif file does not include any time information. All time series information is interpolated dynamically during the run. This is necessary because the time steps in MIKE SHE can dynamically change during the simulation in response to stresses on the system.

The Preprocessed Data Tab is used to display the spatial content of the .fif file.

Before you run your simulation, you should carefully check the preprocessed data for errors. Errors found in the preprocessed data are typically related to incorrectly specified parameters, file names, etc. in the Setup Tab.



#### 4.1.1 Loading your Preprocessed Data



There is only one button in the main dialogue for the processed data dialogue, plus an uneditable text box displaying the current .fif file created by the pre-processor.

**Load** - After you have pre-processed your model setup, and a .fif file is created, you can click on the Load button to load the contents of the .fif file and view the actual model input data.

If the model Setup data has been changed since the last pre-processing, you will get a warning message telling you that the pre-processed data may not match the current setup data.

If you have changed anything in your model setup, and then run the pre-processor again, you must re-load the new .fif file to be able to see the changes in the new .fif file.

#### 4.1.2 Exporting, Saving and Editing the pre-processed data

If you right click on the preprocessed map, you will get a menu, where you can save the current graphic to the clipboard, a .wmf metafile, or a .bmp bitmap file. These options are useful if you want to copy and paste , or import the graphic into a report.

You will also be able to save the pre-processed data to a .dfs2 file. The resulting dfs2 file has the same properties as the model domain and grid. It can be edited directly in the Grid Editor. This option is especially useful if you want to change the preprocessed data. In this case, you can

- 1 preprocess the data,
- 2 save the preprocessed values to a dfs2 file,
- 3 open the dfs2 file in the Grid Editor and edit the values that you want to change, and



4 then use the edited dfs2 file in the Setup tab.

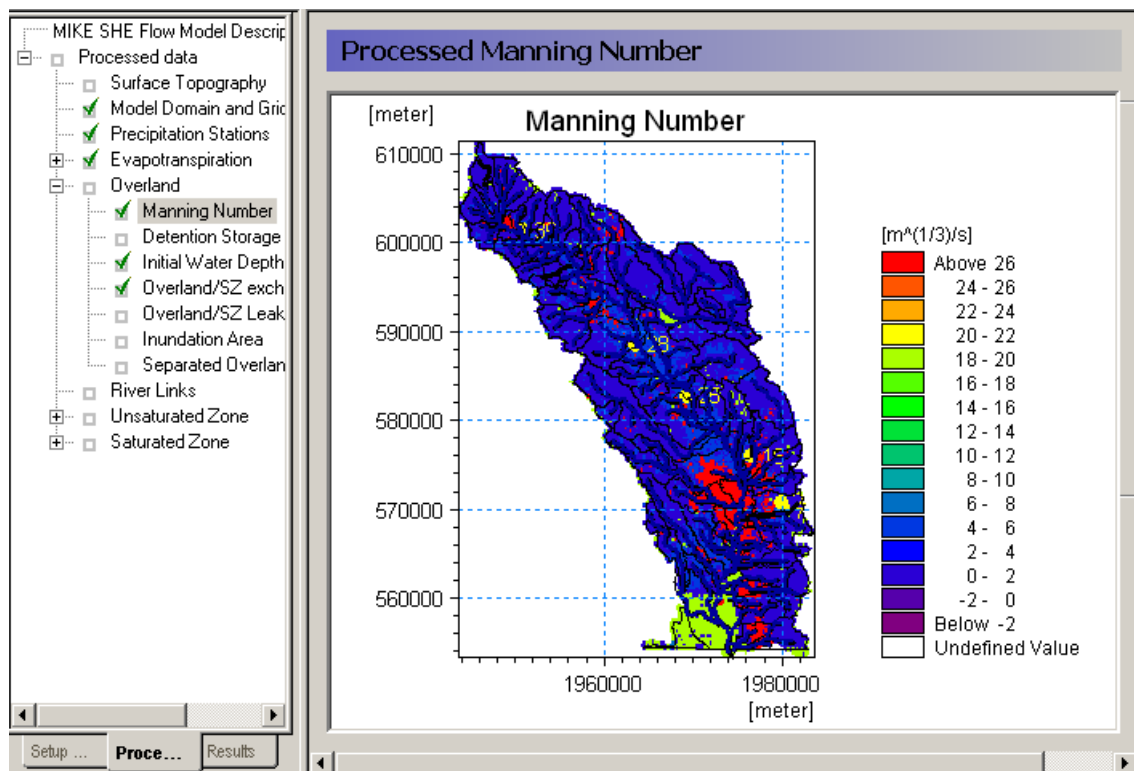
The preprocessed and saved dfs2 file is also useful if you are using a very large shp file or ASCII file in the Setup tab. A large file such as this will be reinterpolated every time you enter the dialogue and during the pre-processing step, which, for a very large file, can be time consuming.

The last option in the right-click-menu is to save the file to a point-theme .shp file, which will contain point values at every node. .shp file, which is useful if you want to create report graphics or otherwise process the data in a GIS.

#### 4.1.3 Updating Overlays in Pre-processed views

The map view displays all of the overlays specified in the Display (V.2 p. 20) item of the Setup Tab. If you want to change the overlays, you can change them in the Setup Tab and they will be automatically updated in the Pre-processed view, without having to re-preprocess your model.

#### 4.1.4 Pre-processed Data Views



Once the pre-preprocessed data in the .fif file has been loaded, then the data tree reflects all the spatial data defined in the model set up tab. In



other words, if the overland flow is not included in the Simulation Specification (V.2 p. 26) dialogue, then the Overland item will not be included in the pre-processed data tree.

### **Model Domain and Grid**

The model domain and grid item displays the grid code values required for the MIKE SHE model. This differs slightly from the Model Domain and Grid (V.2 p. 52) item in the Setup Tab. In the .fif file, all cells outside the model domain are assigned a value of zero, compared to the Setup tab where the cells outside of the model boundary are delete values.

Unlike other data items in the pre-processed tab, you cannot save the pre-processed model domain and grid to a dfs2 file and re-use it in the Setup Tab, because the Model Domain and Grid (V.2 p. 52) item requires delete values outside of the model domain.

### **Precipitation and Evapotranspiration**

The precipitation and evapotranspiration items display the integer station codes for the time series defined in Precipitation Rate (V.2 p. 58) and Reference Evapotranspiration (V.2 p. 79) items in the Setup Tab. The station names are not displayed, so you will have to refer back to the Setup Tab for the station names

However, the .fif file does not include any time information. All time series information is interpolated dynamically during the run. This is necessary because the time steps in MIKE SHE can dynamically change during the simulation in response to stresses on the system.

### **River Links**

The coupling between MIKE 11 and MIKE SHE is made via river links, which are located on the edges that separate adjacent grid cells. The river link network is created by the pre-processor, based on the MIKE 11 coupling reaches. The entire river system is always included in the hydraulic model, but MIKE SHE will only exchange water with the coupling reaches.

The location of each of MIKE SHE river link is determined from the coordinates of the MIKE 11 river points, where the river points include both digitised points and H-points on the specified coupling reaches. Since the MIKE SHE river links are located on the edges between grid cells, the details of the MIKE 11 river geometry can be only partly included in MIKE SHE, depending on the MIKE SHE grid size. The more refined the MIKE SHE grid, the more accurately the river network can be reproduced. This also leads to the restriction that each MIKE SHE grid cell can only

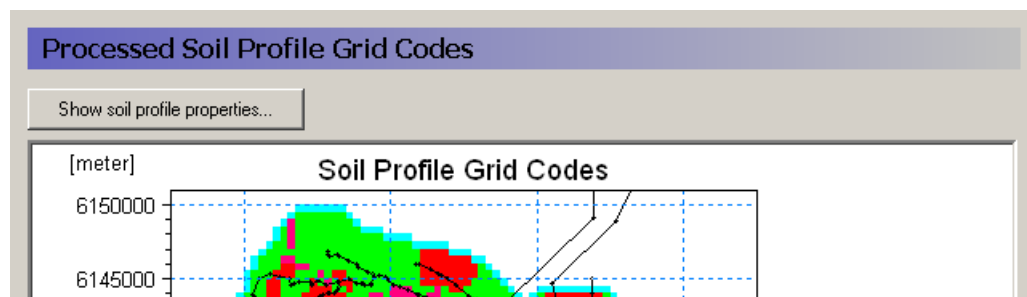




couple to one coupling reach per river link. Thus, if, for example, the distance between coupling reaches is smaller than half a grid cell, you will probably receive an error, as MIKE SHE tries to couple both coupling reaches to the same river link.

The river links are shown on all the maps and the distributed data shown on the River Links map is the Topography.

## UZ Soil Profile Grid Codes

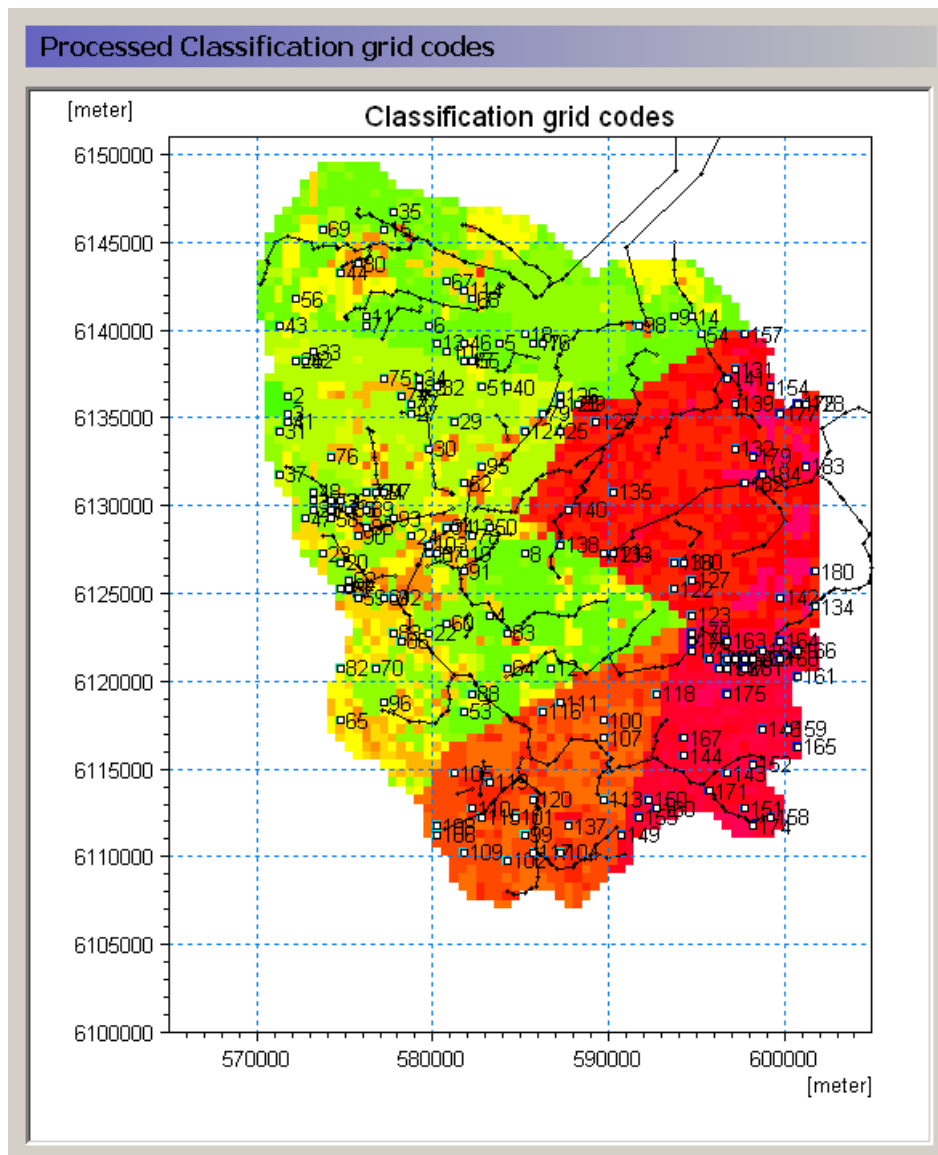


The unsaturated zone is composed of 1D soil columns. If you are using the Richards equation or the Gravity flow method, then these columns consist of a vertical grid with various soil properties. The **Show soil profile properties** button located just above the map allows you to view a summary of the unsaturated zone grid for each cell. If you click on this button, the cursor will change to a target icon. When you click on a particular cell, an ASCII txt file will be created and opened, which contains the summary data.

**Note** that the pre-processor modifies the vertical discretisation wherever the vertical cell size changes. Thus, if you have 10 cells of 20cm thickness, followed by 10 cells of 40cm thickness, the location of the transition will be moved such that the two cells on either side will have an equal thickness. In this case, cells 10 and 11 will both be 15cm.



## UZ Classification Grid Codes



If certain conditions are met, then the flow results for a 1D unsaturated zone column can be applied to columns with similar properties. In this map, each numbered item is a calculation point. The cell with a calculation point is given an integer grid code with a negative value. The flows calculated during the simulation in the cells with the negative code, will be transferred to all the cells with the same positive grid code value. For example, if an UZ recharge to SZ of  $0.5 \text{ m}^3/\text{day}$  is calculated for UZ grid



code -51, then all the SZ cells below the UZ cells with a grid code of +51 will also be given the same recharge.

By just looking at the map it can be difficult to distinguish which calculations are being transferred to which cells. An easier way to look at this is to save the map to a dfs2 file (right click) and then open the file in the Grid Editor, where it is much easier to search for cell numbers.

### **Saturated Zone Items**

The saturated zone items are organized by item with separate maps per layer.

#### **Layer thickness**

Layer thickness is a derived value calculated by subtracting the top and bottom elevations of the layer.

The EUM type is “thickness”, with a MIKE Zero default unit of millimeters. This is not very suitable for geologic layers that can be more than 100m thick. To change the default units, see EUM Data Units (*V.1 p. 349*), find the ‘thickness’ item and change the default unit to meter or feet, as appropriate.

#### **Transmissivity**

Transmissivity is also a derived value calculate by multiplying the thickness by the horizontal hydraulic conductivity.

### **Saturated Zone Drainage**

The rate of saturated zone drainage is controlled by the drain elevation and the drain time constant. However, the destination of the drainage water is controlled by drain codes, which determine if the water flows to a boundary, a local depression, or a river.

#### **SZ Drainage Codes**

The SZ Drainage Codes map is the drainage codes specified in the Drain Codes (*V.2 p. 127*) set up item, interpolated to the model grid.

After the interpolation to the model grid, each active drain cell is mapped to a destination cell. The destination cell is determined from the drain code values and the proximity of rivers and boundaries. Then, whenever drainage is generated in a cell, the drain water will always be routed to the same destination cell.



### **Drainage to local depressions and boundary**

All cells with the same positive code are drained to the cell with the same numeric negative code.

### **Drainage to river**

All cells with the same positive code are drained to the cell with the same numeric negative code.

## **4.2 *Running your Model***

### **4.2.1 *Running MIKE SHE from a Batch File***

A 'batch' file contains native DOS commands in a programming structure. When executed each of the DOS commands in the batch file is executed sequentially. Since, most MIKE Zero and MIKE SHE programs can be executed in this way, a properly constructed batch file allows you to run multiple models sequentially when you are not at the computer, such as over night.

Basically, to run MIKE SHE in batch mode, you must

- 1 Setup the different models with different names using the Setup Editor
- 2 Create a .BAT file containing the DOS commands to run the models
- 3 Run the .BAT file and analyse the results using the standard MIKE Zero analysis tools (e.g. the Results Viewer)

### **Setup the different models**

Your original model can be saved to a new name and the necessary changes made in the new set up. We highly recommended that you create and set up the different models in the MIKE SHE Setup Editor. In principle, you could edit the .SHE file, which is a text file containing all of the information on the model set up, but the file is typically very large and confusing, and the format of this file must be preserved exactly.

### **Create the batch file**

To create a batch file, you must create a text file with the extension .BAT. Then add the DOS commands in the order that you would like them executed. But, before you can run the MIKE SHE executables, you must add the MIKE SHE installation directory to your PATH variable. The default installation directory is:



```
C:\Program Files\DHI\MIKEZero\bin\
```

The DOS command to add the default path to the PATH variable is:

```
Set PATH=%PATH%,C:\Program Files\DHI\MIKEZero\bin\
```

To run MIKE SHE from the batch file you must add the following two DOS command lines after the PATH statement above:

```
MSHE_PreProcessor MyModel.she  
MSHE_watermovement MyModel.she
```

The above two lines will run both the preprocessor and the water movement engine separately. If you want to run them together, then you can replace the two lines with

```
MSHE_Simulation MyModel.she
```

The examples above will run silently. That is, no progress information will be displayed. If you want to display progress information, then you should use the MzLaunch utility. Using

```
MzLaunch.exe MyModel.she -e MSHE_Simulation
```

will leave the MzLaunch utility open when the simulation finishes, whereas

```
MzLaunch.exe MyModel.she -e MSHE_Simulation -exit
```

will close the MzLaunch utility when the simulation finishes.

### Analyse the Results

The MSHE\_watermovement.exe program automatically generates all of the output asked for in the Setup Editor. Thus, to look at your output, you only need to open the model at look at your results in the normal way.

If you want to run the water balance program, which is described in the Using the Water Balance Tool chapter, you can add the following lines to you batch file:

```
MSHE_Wbl_Ex.exe //apv My_WB_areas.WBL
```

```
MSHE_Wbl_Post.exe //apv My_WB_areas.WBL 1
```

```
MSHE_Wbl_Post.exe //apv My_WB_areas.WBL 2
```



In the above, the first command runs the Extraction phase of the water balance utility, while the subsequent commands run the Post-processing items in the water balance file. The number after the water balance file name indicates which Post-processing item to run. Post-processing steps cannot be executed before an Extraction step but only one Extraction step needs to be run for each water balance utility file.

#### **4.2.2 Controlling the Time Steps**

Each of the main hydrologic components in MIKE SHE run with independent time steps. Although, the time step control is automatically controlled, whenever possible, MIKE SHE will run with the maximum allowed time steps.

**Note** In the 2007 Release the MIKE 11 time step is no longer specified in MIKE SHE.

The component time steps are independent, but they must meet to exchange flows, which leads to some restrictions on the specification of the maximum allowed time steps.

- If MIKE 11 is running with a constant time step, then the Max allowed Overland (OL) time step must be a multiple of the MIKE 11 constant time step. If MIKE 11 is running with a variable time step, then the actual OL time step will be truncated to match up with the nearest MIKE 11 time step.
- The Max allowed UZ time step must be an even multiple of the Max allowed OL time step, and
- The Max allowed SZ time step must be an even multiple of the Max allowed UZ time step.

Thus, the overland time step is always less than or equal to the UZ time step and the UZ time step is always less than or equal to the SZ time step.

If you are using the implicit solver for overland flow, then a maximum OL time step equal to the UZ time step often works. However, if you are using the explicit solver for overland flow, then a much smaller maximum time step is necessary, such as the default value of 0.5 hours.

If the unsaturated zone is included in your simulation and you are using the Richards equation or Gravity Flow methods, then the maximum UZ time step is typically around 2 hours. Otherwise, a maximum time step equal to the SZ time step often works.



Groundwater levels react much slower than the other flow components. So, a maximum SZ time step of 24 or 48 hours is typical, unless your model is a local-scale model with rapid groundwater-surface water reactions.

### **Precipitation-dependent time step control**

Periods of heavy rainfall can lead to numerical instabilities if the time step is too long. To reduce the numerical instabilities, a time step control has been introduced on the precipitation and infiltration components. You will notice the effect of these factors during the simulation by suddenly seeing very small time steps during storm events.

The parameters controlling the time step adjustment are in the Time Step Control (V.2 p. 30) dialogue. In particular, the following three parameters control the time step during rainfall events:

**Max precipitation depth per time step** If the total amount of precipitation [mm] in the current time step exceeds this amount, the time step will be reduced by the increment rate. Then the precipitation time series will be resampled to see if the max precipitation depth criteria has been met. If it has not been met, the process will be repeated with progressively smaller time steps until the precipitation criteria is satisfied. Multiple sampling is important in the case where the precipitation time series is more detailed than the time step length. However, the criteria can lead to very short time steps during short term high intensity events. For example, if your model is running with maximum time steps of say 6 hours, but your precipitation time series is one hour, a high intensity one hour event could lead to time steps of a few minutes during that one hour event.

**Max infiltration amount per time step** If the total amount of infiltration due to ponded water [mm] in the current time step exceeds this amount, the time step will be reduced by the increment rate. Then the infiltration will be recalculated. If the infiltration criteria is still not met, the infiltration will be recalculated with progressively smaller time steps until the infiltration criteria is satisfied.

**Input precipitation rate requiring its own time step** If the amount of precipitation [mm] divided by the time step length [hr] in the current time step exceeds this amount, the time step will be reduced by the increment rate until this criteria is met. That is, the precipitation time series will be resampled with progressively smaller time steps until the precipitation rate criteria is satisfied. Multiple sampling is important in the case where the precipitation time series is more detailed than the time step length. However, the criteria can lead to very short time steps



during short term high intensity events. For example, if your model is running with maximum time steps of say 6 hours, but your precipitation time series is one hour, a high intensity one hour event could lead to time steps of a few minutes during that one hour event.

If your model does not include the unsaturated zone, or if you are using the 2-Layer water balance method, then you can set these conditions up by a factor of 10 or more. However, if you are using the Richards equation method, then you may have to reduce these factors to achieve a stable solution.

### **Actual time step for the different components**

As outlined above the overland time step is always less than or equal to the UZ time step and the UZ time step is always less than or equal to the SZ time step. However, the exchanges are only made at a common time step boundary. This means that if one of the time steps is changed, then all of the time steps must change accordingly. To ensure that the time steps always meet, the initial ratios in the maximum time steps specified in this dialogue are maintained.

After a reduction in time step, the subsequent time step will be increased by

$$timestep = timestep \times (1 + IncrementRate) \quad (4.1)$$

until the maximum allowed time step is reached.

### **Relationship to Storing Time Steps**

The Storing Time Step specified in the Detailed time series output (V.2 p. 138) dialogue, must also match up with maximum time steps. Thus,

- The OL storing time step must be an integer multiple of the Max UZ time step,
- The UZ storing time step must be an integer multiple of the Max UZ time step,
- The SZ storing time step must be an integer multiple of the Max SZ time step,
- The SZ Flow storing time step must be an integer multiple of the Max SZ time step, and





- The Hot start storing time step must be an integer multiple of the maximum of all the storing time steps (usually the SZ Flow storing time step)

For example, if the Maximum allowed SZ time step is 24 hrs, then the SZ Storing Time Step can only be a multiple of 24 hours (i.e. 24, 48, 72 hours, etc.)

#### 4.2.3 *Speeding up your simulation*

In most cases, the best way to speed up your model is to make it simpler. You should look very carefully at your model and ask yourself the following questions, for example:

- **Do you really need a fine discretisation during calibration?** - A coarser grid may allow you to do many more calibration runs. Then when the model is calibrated, you can refine the grid for the final simulations - but remember to check you calibration first.
- **Do you really need the Richards equation for unsaturated flow?** - For regional models, the two layer water balance method is usually sufficient, which is very fast. The gravity flow method is also, typically 2-5 times faster than the Richards equation method. Again during the calibration it can be a good idea to use one of the simpler methods and the more detailed method for the final simulations.
- **Is your MIKE 11 simulation too detailed?** - If your MIKE 11 cross-sections are too close together, MIKE 11 will run with a very short time step. Regional models can often be run with the simple routing methods in MIKE 11, which are very fast.

If your simulation is still too slow, then the section Hardware Requirements (*V.1 p. 23*) contains some information that might be of interest.





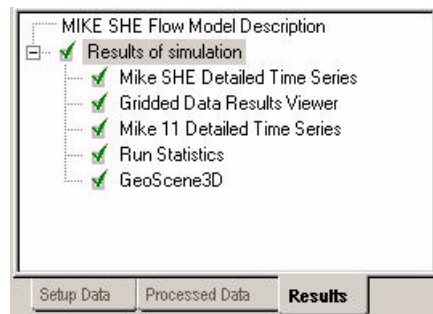
*Phi Software*

**WORKING WITH YOUR RESULTS**





## 5 THE RESULTS TAB



All the simulation results are collected in the Results tab. This includes Detailed time series output for both MIKE SHE and MIKE 11, as well as Grid series output for MIKE SHE.

A Run Statistics tool is available for helping you assimilate the calibration statistics for each of the detailed time series plots.

A link to the GeoScene3D program is also included, where you can visualize your results in a dynamic 3D environment.

### 5.1 Linking 2005 Results in a 2007 .she Set Up File

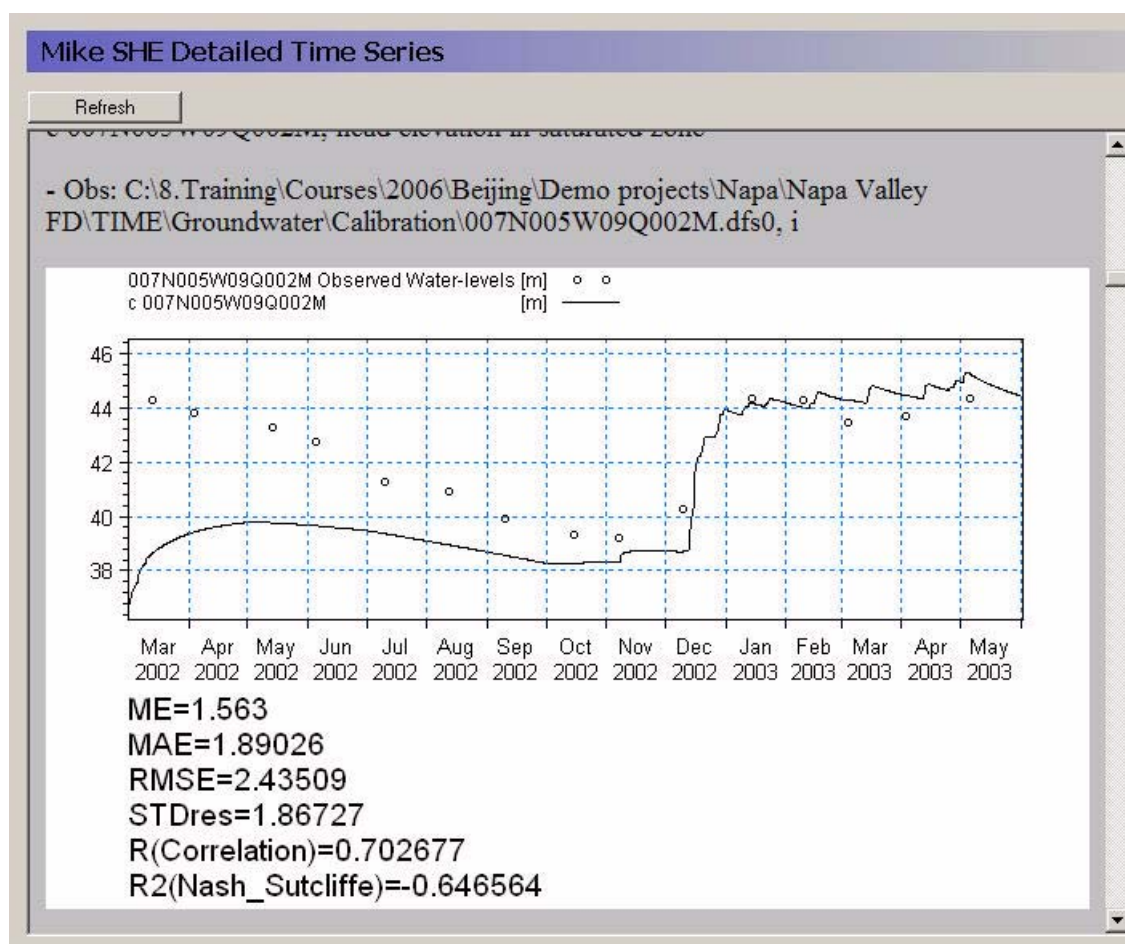
The 2007 MIKE Zero project template places all Results from your project in a common Results folder. This is different from the 2005 Release, where all of your MIKE SHE results were placed in a folder under your project with the same name as the project. This means that when you open your 2005 project in MIKE SHE you may not be able to access your results in the Results Tab.

To link the Results Tab to your existing results, without rerunning your model, you need to uncheck the Default Output folder option in the Stor-



ing of results dialogue. Then use the browse button to specify the directory where your results are stored, as shown below

## 5.2 Detailed Time Series Results



The MIKE SHE Detailed time series tab includes an HTML plot of each point selected in the Setup Editor. The HTML plots are updated during the

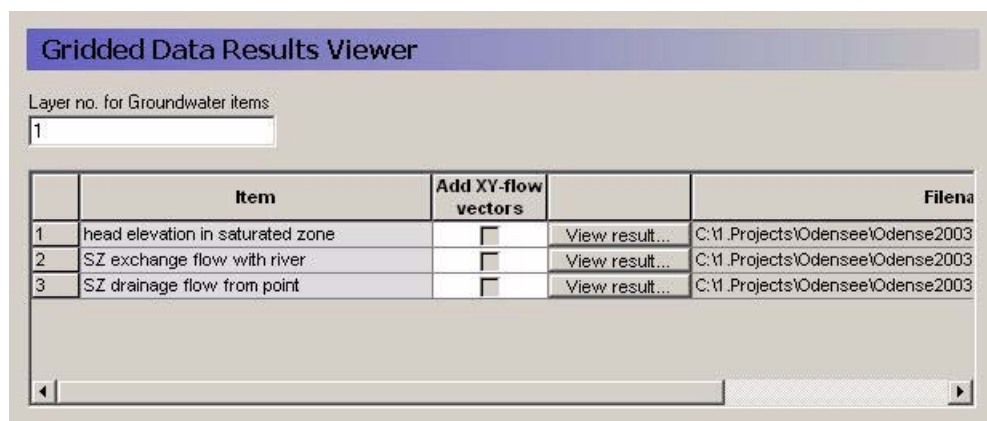


simulation whenever you enter the view. Alternatively, you can select the Refresh button to refresh the plot.

Note that the HTML plot is regenerated every time you enter the view. So, if you have a lot of plots and a long simulation, then the regeneration can take a long time.

For information on the statistics see *Statistic Calculations (V.2 p. 161)*.

### 5.3 Gridded Results



Gridded data results for MIKE SHE can be viewed by selecting the Gridded Data Results Viewer item on the Results tab. The table is a list of all gridded data saved during a MIKE SHE simulation. The items in this list originate from the list of items selected in the Grid series output (*V.2 p. 143*) dialogue from the Setup tab.

Clicking on the View result button will open the Results Viewer to the current item. All overlays from MIKE SHE (e.g. shape files, images, and grid files) will be transferred as overlays to the result view. However, the MIKE 11 river network is not transferred as an overlay.

For 3D SZ data files, the layer number can be specified at the top of the table. However, the layer number can be changed from within the Results Viewer (see *Changing to a different SZ layer (V.1 p. 115)*) By default the top layer is displayed.

Vectors can be added to the SZ plots of results, by checking the *Add X-Y flow vectors* checkbox. These vectors are calculated based on the *Groundwater flow in X-direction* and *Groundwater flow in Y-direction* data types if they were saved during the simulation.



In the current version, velocity vectors cannot be added for overland flow output.

### **5.3.1 “The Result Viewer setup file already exists” warning**

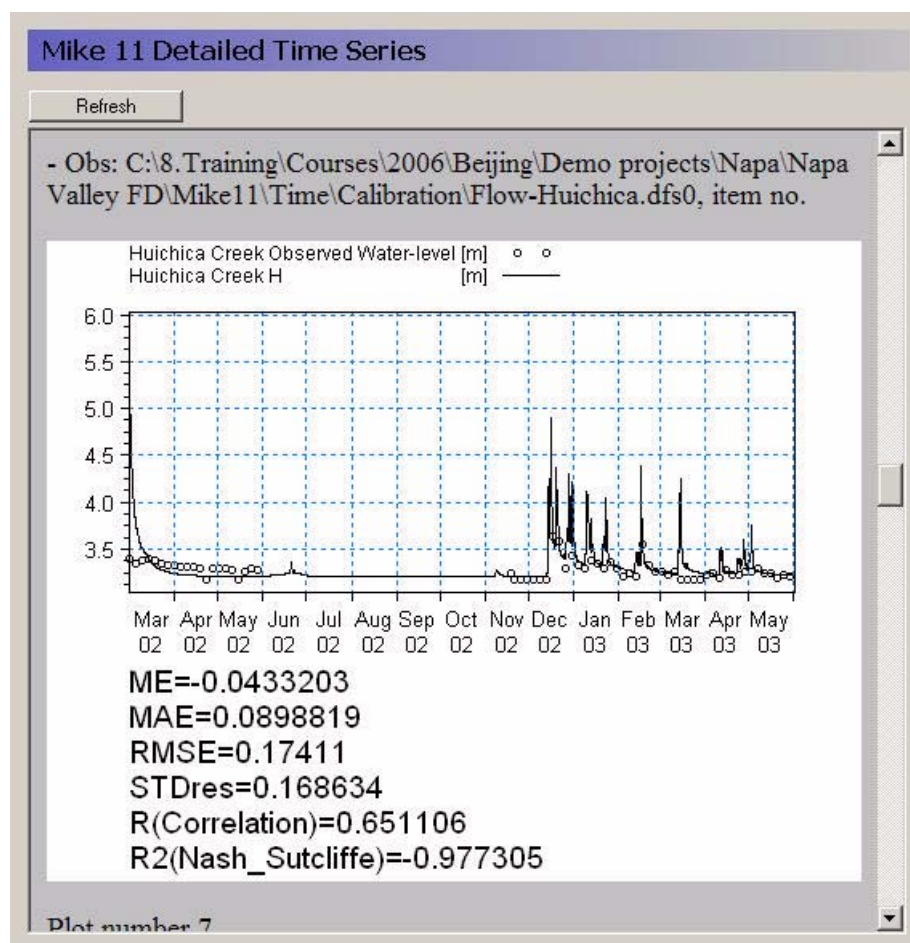
When the Result Viewer opens one of the items in the table, it creates a setup file for the particular view with the extension .rev. The name of the current setup file is displayed in the title bar of the dialogue. Initially, the .rev file includes only the default view settings and the overlay information from MIKE SHE. However, if you make changes to the view, such as changes the way contours are displayed, then when you close the view, you will be asked if you want to save your changes.

Thus, the next time you open the item in the table, you will be asked if you want to overwrite the existing .rev file. If you click on “Yes”, then a new .rev file will be created. If you click on “No”, then your previous settings will be re-loaded. This is a convenient way to set up the contouring, legend, etc., the way you want and then re-use the settings.





## 5.4 MIKE 11 Detailed Time Series



The MIKE 11 Detailed time series tab includes an HTML plot of each point selected in the Setup Editor. The HTML plots are updated during the simulation whenever you enter the view. Alternatively, you can select the Refresh button to refresh the plot.

Note that the HTML plot is regenerated every time you enter the view. So, if you have a lot of plots and a long simulation, then the regeneration can take a long time.

For information on the statistics see *Statistic Calculations (V.2 p. 161)*.



## 5.5 Run Statistics

Run statistics can be generated in HTML format for a MIKE SHE simulation. The run statistics table information can be copied and pasted directly into any word processing program, such as Microsoft Word, or spreadsheet, such as Microsoft Excel. The Run Statistics HTML document includes MIKE SHE and MIKE 11 results for all items included in the MIKE SHE and MIKE 11 detailed time series sections that also include observation data.

To calculate Run Statistics for a simulation, navigate to the Results Tab and the Run Statistics item on the menu tree. Press the Generate Statistics button on the Run Statistics window to perform the statistical calculations. For some simulations with long simulation periods and/or a lot of calibration data it can take a while to generate the run statistics.

After successful completion of the Generate Statistics phase, the Run Statistics HTML document will be displayed in the window on the Run Statistics page (see below).

Run Statistics

Refresh

Generate Statistics

Start Date:  
2002/03/01 02:00

End Date:  
2003/06/02 02:00

| Name             | Data type                        | X            | Y      | Layer | ME       | MAE     | RMSE    | STDres   | R<br>(Correlation) |
|------------------|----------------------------------|--------------|--------|-------|----------|---------|---------|----------|--------------------|
| c005N003W06R001M | head elevation in saturated zone | 1.97985e+006 | 570854 | 2     | 61.5017  | 61.5017 | 62.2307 | 9.49734  | -1                 |
| c005N003W08E001M | head elevation in saturated zone | 1.98032e+006 | 569892 | 2     | 113.682  | 113.682 | 113.989 | 8.37141  | -1                 |
| c-----           | head elevation in                | 1.97639e+006 | 575988 | 1     | -6.69369 | 6.69369 | 6.69454 | 0.106626 | 1                  |

Similar to the detailed time series output, the Run Statistics can be viewed during a simulation. Press the Refresh button on the Run Statistics page to update the Run Statistics using the most recent model results during a simulation

For information on the statistics see *Statistic Calculations (V.2 p. 161)*.



### 5.5.1 **Shape file output for run statistics**

A shape file of statistics is also generated when the html document is generated. The shape file contains all of the information contained in the HTML document and can be used to generate maps of model errors that can be used to evaluate spatial bias. The shape file is created in the simulation directory and is named *ProjectName\_Stat.shp* where *ProjectName* is the name of the \*.she file for the simulation. Note: the Run Statistics shape file does not have a projection file associated with it and this file should be created using standard ArcGIS methods.

The statistics contained in the HTML document and the shape file are calculated using the same methods used to calculate statistics for the detailed time series output. The reader is referred to the Detailed Time Series Output section for more information on how the statistics are calculated.





## 6 OUTPUT ITEMS

The available output items for gridded data and time series data are listed in Table 6.1 and Table 6.2. Table 6.2 lists a number of additional output items, such as the number of solver iterations, that can only be displayed as a time series.

The list of available output items depends on the processes selected in the Simulation Specification dialogue. Thus, for example, the items for Overland Flow only appear when Overland flow is being calculated.

Some of the items in the list are calculated as part of another process. For example, the depth of overland water is calculated based on seepage to and from the groundwater and as part of the MIKE 11 surface water calculations, even if the overland flow is not directly simulated.

Furthermore, some of the output items require that more than one process be simulated. For example, the leaf area index is only available if both evapotranspiration and unsaturated flow are calculated.

Also, some of the items are input items, such as precipitation, which is usually input as a time series for several polygons.

In Table 6.1 and Table 6.2, the Data Type Code is used only when importing time series items into the Detailed time series output (V.2 p. 138) dialogue.

**Table 6.1** Available output items for gridded data and time series.

**–Key to symbols**

- ET - Evapotranspiration
- OL - Finite Difference Overland Flow
- SubOL - Sub-catchment based Overland Flow
- UZ - Richards or Gravity Unsaturated flow,
- 2LUZ - 2-Layer Unsaturated Water Balance
- SZ - Finite Difference Saturated Zone flow,
- LR - Linear Reservoir groundwater
- AD - Advection Dispersion (Water Quality)
- PT - Particle Tracking

| Data Type Code | Output Item                            | Appears when this process is selected |         |
|----------------|--|---------------------------------------|---------|
| 10             | precipitation rate                     | Always                                |         |
| 128            | average water content in the root zone | UZ+ET                                 | 2LUZ+ET |
| 11             | rooting depth                          | UZ+ET                                 | 2LUZ+ET |



**Table 6.1** Available output items for gridded data and time series.

**–Key to symbols**

–ET - Evapotranspiration

–OL - Finite Difference Overland Flow

–SubOL - Sub-catchment based Overland Flow

–UZ - Richards or Gravity Unsaturated flow,

–2LUZ - 2-Layer Unsaturated Water Balance

–SZ - Finite Difference Saturated Zone flow,

–LR - Linear Reservoir groundwater

–AD - Advection Dispersion (Water Quality)

–PT - Particle Tracking

| Data Type Code | Output Item  | Appears when this process is selected |    |                        |
|----------------|--|---------------------------------------|----|------------------------|
|                |  |                                       |    |                        |
| 12             | leaf area index  | UZ+ET                                 |    | 2LUZ+ET                |
| 182            | crop coefficient   | UZ+ET                                 |    | 2LUZ+ET                |
| 15             | actual evapotranspiration  | UZ+ET                                 |    | 2LUZ+ET                |
| 16             | actual transpiration   | UZ+ET                                 |    | 2LUZ+ET                |
| 13             | actual soil evaporation  | UZ+ET                                 |    |                        |
| 17             | actual evaporation from interception   | UZ+ET                                 |    | 2LUZ+ET                |
| 18             | actual evaporation from ponded water   | UZ+ET                                 |    | 2LUZ+ET                |
| 19             | canopy interception storage  | UZ+ET                                 |    | 2LUZ+ET                |
| 14             | evapotranspiration from SZ   | SZ+UZ+ET                              |    | SZ+2LUZ+ET             |
| 100            | snow storage   | UZ+ET+ snowmelt                       |    | 2LUZ+ET+ SM            |
| 99             | sublimation from snow  | ET+ snowmelt                          |    |                        |
| 61             | depth of overland water  | OL                                    | SZ | SubOL                  |
| 58             | overland flow in x-direction<br>(this is the flow across the boundary from cell <sub>i</sub> to cell <sub>i+1</sub> in volume/time e.g. m <sup>3</sup> /s) | OL                                    |    |                        |
| 59             | overland flow in y-direction<br>(this is the flow across the boundary from cell <sub>i</sub> to cell <sub>i+1</sub> in volume/time e.g. m <sup>3</sup> /s) | OL                                    |    |                        |
| 62             | paved area drainage to river or MOUSE  | OL+M11+ Drainage+ Paved               |    | MOUSE+ Drainage+ Paved |



**Table 6.1** Available output items for gridded data and time series.

**–Key to symbols**

- ET - Evapotranspiration
- OL - Finite Difference Overland Flow
- SubOL - Sub-catchment based Overland Flow
- UZ - Richards or Gravity Unsaturated flow,
- 2LUZ - 2-Layer Unsaturated Water Balance
- SZ - Finite Difference Saturated Zone flow,
- LR - Linear Reservoir groundwater
- AD - Advection Dispersion (Water Quality)
- PT - Particle Tracking

| Data Type Code | Output Item  | Appears when this process is selected |      |         |    |
|----------------|--|---------------------------------------|------|---------|----|
| 141            | Water content in root zone (2-layer UZ)                      | 2LUZ                                  |      |         |    |
| 142            | Water content below root zone (2-layer UZ)                   | 2LUZ                                  |      |         |    |
| 143            | Maximum water content (2-layer UZ)                           | 2LUZ                                  |      |         |    |
| 144            | Minimum water content (2-layer UZ)                           | 2LUZ                                  |      |         |    |
| 121            | infiltration to UZ (negative)                                | UZ                                    | 2LUZ | SZ      | LR |
| 122            | exchange from UZ to SZ (negative)                            | UZ                                    | 2LUZ | SZ      | LR |
| 57             | UZ deficit   | UZ                                    | 2LUZ |         |    |
| 37             | average soil moisture content in top 5 compartments          | LR+UZ                                 |      |         |    |
| 119            | rate of change in UZ storage                                 | UZ                                    |      |         |    |
| 123            | epsilon calculated in UZ                                     | UZ                                    | 2LUZ |         |    |
| 120            | accumulated error in UZ (water balance in the UZ cells only) | UZ                                    |      |         |    |
| 45             | groundwater feedback to the unsaturated zone                 | LR+UZ                                 |      | LR+2LUZ |    |
| 117            | unsaturated zone flow  | UZ                                    |      |         |    |
| 118            | water content in unsaturated zone                            | UZ                                    |      |         |    |
| 159            | pressure head in unsaturated zone                            | UZ                                    |      |         |    |
| 129            | root water uptake  | UZ+ET                                 |      |         |    |
| 20             | irrigation: actual water content in root zone                | UZ+ET+Irrigation                      |      |         |    |
| 135            | irrigation: soil moisture deficit in root zone               | UZ+ET+Irrigation                      |      |         |    |



**Table 6.1** Available output items for gridded data and time series.

**–Key to symbols**

- ET - Evapotranspiration
- OL - Finite Difference Overland Flow
- SubOL - Sub-catchment based Overland Flow
- UZ - Richards or Gravity Unsaturated flow,
- 2LUZ - 2-Layer Unsaturated Water Balance
- SZ - Finite Difference Saturated Zone flow,
- LR - Linear Reservoir groundwater
- AD - Advection Dispersion (Water Quality)
- PT - Particle Tracking

| Data Type Code | Output Item  | Appears when this process is selected |
|----------------|--|---------------------------------------|
| 21             | total irrigation   | UZ+ET+Irrigation                      |
| 26             | irrigation from river  | M11+UZ+ET+Irrigation                  |
| 28             | irrigation from wells  | SZ+UZ+ET+Irrigation                   |
| 22             | irrigation from external source  | UZ+ET+Irrigation                      |
| 23             | irrigation index   | UZ+ET+Irrigation                      |
| 24             | irrigation shortage  | UZ+ET+Irrigation                      |
| 25             | irrigation total demand  | UZ+ET+Irrigation                      |
| 153            | sprinkler irrigation   | UZ+ET+Irrigation                      |
| 154            | drip and sheet irrigation  | UZ+ET+Irrigation                      |
| 27             | ground water extraction for irrigation   | SZ+UZ+ET+Irrigation                   |
| 106            | depth to phreatic surface (negative)   | SZ                                    |
| 101            | head elevation in saturated zone   | SZ                                    |
| 107            | seepage flow SZ -overland<br>(the flow up from SZ onto the topography)           | SZ                                    |
| 108            | seepage flow overland - SZ (negative)<br>(the flow down into the saturated zone) | SZ                                    |
| 113            | 3D UZ recharge to SZ (negative)  | SZ+NegPrec                            |
| 102            | groundwater flow in x-direction<br>(a flow rate, e.g. in [m <sup>3</sup> /s])    | SZ                                    |
| 103            | groundwater flow in y-direction<br>(a flow rate, e.g. in [m <sup>3</sup> /s])    | SZ                                    |





**Table 6.1** Available output items for gridded data and time series.

**–Key to symbols**

- ET - Evapotranspiration
- OL - Finite Difference Overland Flow
- SubOL - Sub-catchment based Overland Flow
- UZ - Richards or Gravity Unsaturated flow,
- 2LUZ - 2-Layer Unsaturated Water Balance
- SZ - Finite Difference Saturated Zone flow,
- LR - Linear Reservoir groundwater
- AD - Advection Dispersion (Water Quality)
- PT - Particle Tracking

| Data Type Code | Output Item   | Appears when this process is selected |
|----------------|---|---------------------------------------|
| 104            | groundwater flow in z-direction<br>(a vertical darcy flow rate, e.g. in [mm/day])   | SZ                                    |
|                | SZ head elevation stored with SZ flows<br>(for SZ cross-sections in Results Viewer) | SZ                                    |
| 109            | groundwater extraction  | SZ+Extraction                         |
| 115            | SZ exchange flow with river   | SZ+River                              |
| 112            | SZ drainage flow from point   | SZ+Drainage                           |
| 105            | SZ flow to general head boundary  | SZ+GHB                                |
| 216            | Overland concentration  | OC                                    |
| 217            | Overland sorbed concentration   | OC                                    |
| 218            | Overland mass/area  | OC                                    |
| 219            | Air temperature   | OC                                    |
| 220            | UZ concentration (matrix phase)   | UZ                                    |
| 221            | UZ sorbed concentration (matrix phase)  | UZ                                    |
| 222            | UZ concentration (macropore phase)  | UZ                                    |
| 223            | UZ sorbed concentration (macropore phase)   | UZ                                    |
| 224            | UZ mass flux (matrix phase)   | UZ                                    |
| 225            | UZ mass flux (macropore phase)  | UZ                                    |
| 226            | UZ soil temperature   | UZ                                    |
| 227            | SZ concentration (mobile phase)   | SZ                                    |
| 228            | SZ sorbed concentration (mobile phase)  | SZ                                    |



**Table 6.1** Available output items for gridded data and time series.

**–Key to symbols**

- ET - Evapotranspiration
- OL - Finite Difference Overland Flow
- SubOL - Sub-catchment based Overland Flow
- UZ - Richards or Gravity Unsaturated flow,
- 2LUZ - 2-Layer Unsaturated Water Balance
- SZ - Finite Difference Saturated Zone flow,
- LR - Linear Reservoir groundwater
- AD - Advection Dispersion (Water Quality)
- PT - Particle Tracking

| Data Type Code | Output Item   | Appears when this process is selected |
|----------------|---|---------------------------------------|
| 229            | SZ concentration (immobile phase)                   | SZ                                    |
| 230            | SZ sorbed concentration (immobile phase)            | SZ                                    |
| 231            | SZ soil temperature                                 | SZ                                    |
| 232            | SZ porosity   | SZ                                    |
| 233            | Number of particles                                 | SZ                                    |
| 234            | Number of registered particles                      | SZ                                    |
| 235            | Most recent registration zone code                  | SZ                                    |
| 236            | Average age   | SZ                                    |
| 237            | Average transport time to nearest registration cell | SZ                                    |



Table 6.2 Additional output items for time series.

**–Key to symbols**

–ET - Evapotranspiration

–OL - Finite Difference Overland Flow

–SubOL - Sub-catchment based Overland Flow

–UZ - Richards or Gravity Unsaturated flow,

–2LUZ - 2-Layer Unsaturated Water Balance

–SZ - Finite Difference Saturated Zone flow,

–LR - Linear Reservoir groundwater

–AD - Advection Dispersion

–PT - Particle Tracking

| Data Type Code | Output Item  | Appears when this process is selected |         |
|----------------|--|---------------------------------------|---------|
| 145            | SimStatus: Basic time step length                      | UZ                                    | OL      |
| 146            | SimStatus: SZ time step length                         | SZ                                    |         |
| 147            | SimStatus: No. of SZ iterations / time step            | SZ                                    |         |
| 148            | SimStatus: Avg. no. UZ iterations / column / time step | UZ                                    |         |
| 149            | SimStatus: No. of Overland iterations per time step    | OL                                    |         |
| 29             | recharge to interflow reservoirs                       | LR                                    |         |
| 30             | interflow from interflow reservoirs                    | LR                                    |         |
| 31             | percolation from interflow reservoirs                  | LR                                    |         |
| 32             | interflow reservoir storage                            | LR                                    |         |
| 33             | change in interflow reservoir storage                  | LR                                    |         |
| 34             | inflow to baseflow reservoir                           | LR                                    |         |
| 211            | dead zone inflow to baseflow reservoir                 | LR                                    |         |
| 35             | baseflow from baseflow reservoir                       | LR                                    |         |
| 36             | groundwater feedback from baseflow reservoir           | LR+UZ                                 | LR+2LUZ |
| 44             | pumping from baseflow reservoir                        | LR                                    |         |
| 46             | storage in baseflow reservoir                          | LR                                    |         |
| 212            | dead zone storage in baseflow reservoir                | LR                                    |         |
| 38             | change in subcatchment storage in baseflow reservoir   | LR                                    |         |
| 213            | change in dead zone storage in baseflow reservoir      | LR                                    |         |
| 155            | simple overland water depth                            | SubOL                                 |         |



Table 6.2 Additional output items for time series.

**–Key to symbols***–ET - Evapotranspiration**–OL - Finite Difference Overland Flow**–SubOL - Sub-catchment based Overland Flow**–UZ - Richards or Gravity Unsaturated flow,**–2LUZ - 2-Layer Unsaturated Water Balance**–SZ - Finite Difference Saturated Zone flow,**–LR - Linear Reservoir groundwater**–AD - Advection Dispersion**–PT - Particle Tracking*

| <b>Data Type Code</b> | <b>Output Item</b>                              | <b>Appears when this process is selected</b> |
|-----------------------|---|--|
| 156                   | simple overland exchange to lower zone or river | SubOL  |
| 157                   | simple overland recharge                        | SubOL  |



## 7 THE RESULTS VIEWER

### 7.1 Toolbars



Many of the functions in the Results Viewer are the same as those available in other DHI software tools (e.g., 2D Grid Editor). Additional tools available in the result viewer are summarized in Table 7.1.

Table 7.1 Description of Result Viewer tools











| Button  | Name            | Description   |
|---|-----------------|---|
|    | Rewind          | Rewinds result files to first time step   |
|  | Previous Step   | Rewinds result files to the previous time step.   |
|  | Video Reverse   | Generates an avi file from the current time step to the first time step   |
|  | Play Reverse    | Plays result files from the current time step to the first time step. Identical to Video Reverse except an avi file is not generated. |
|  | Stop Animation  | Stops forward and reverse playing of result files and creation of avi files   |
|  | Play Forward    | Plays result files from the current time step to the last time step. Identical to Video Forward except an avi file is not generated.  |
|  | Video Forward   | Generates an avi file from the current time step to the last time step  |
|  | Next Step       | Advances result files from the current time step to the next time step  |
|  | Wind            | Advances results files to the last time step  |
|  | Go to time step | Rewinds or advances result files to the specified time step   |





Table 7.1 Description of Result Viewer tools

| Button | Name                    | Description   |
|--------|-------------------------|---|
|        | Time step               | Change the time step used by the result viewer. The time step can be less than or greater than the result file time step  |
|        | Default                 | Default extraction tool.  |
|        | Time Series extractor   | Tool to extract time series data from result files. Multiple time series can be extracted by holding down the Ctrl key while left-clicking. A single extraction or the last multiple extraction is selected using a double left-click. See Displaying a time series at a point (V.1 p. 108) |
|        | Profile extractor       | Tool to extract vertical profiles (cross-sections) from 3D result files. Vertices of a profile line are specified with a single left click and the profile line is closed with a double left-click. See Saturated Zone Cross-section Plots (V.1 p. 111)                                     |
|        | Cross-section extractor | Tool to extract cross-sections of MIKE 11 results at H-points. Additional information is given below  |
|        | UZ Plot extractor       | Tool to extract a UZ plot of the water content in the unsaturated zone. This tool generates a plot of water content versus depth with time. This tool can only be used on one cell at a time. A cell is selected by double left-click. Additional information is given below.               |



Table 7.1 Description of Result Viewer tools

| Button  | Name            | Description   |
|---|-----------------|---|
|  | UZ Scatter plot | Limits displays of results to unsaturated zone calculation cells. This button is only activated if unsaturated zone data is displayed in the result viewer. Additional information is given below.  |
|  | UZ Filled Plot  | Displays interpolated unsaturated zone results in non-calculation cells and unsaturated zone results in calculation cells. This button is only activated if unsaturated zone data is displayed in the result viewer. Additional information is given below. |

## 7.2 Modifying the plot

When the Results viewer is opened from MIKE SHE, a default plot is created. However, in many cases, you will want to edit these plot settings. Typical changes fall into four broad categories:

- Adding additional result files and overlays (V.1 p. 95)
- Adding or modifying vectors (V.1 p. 98)
- Changing the shading or contour settings (V.1 p. 100)
- Changing the legend and colour scale (V.1 p. 105)

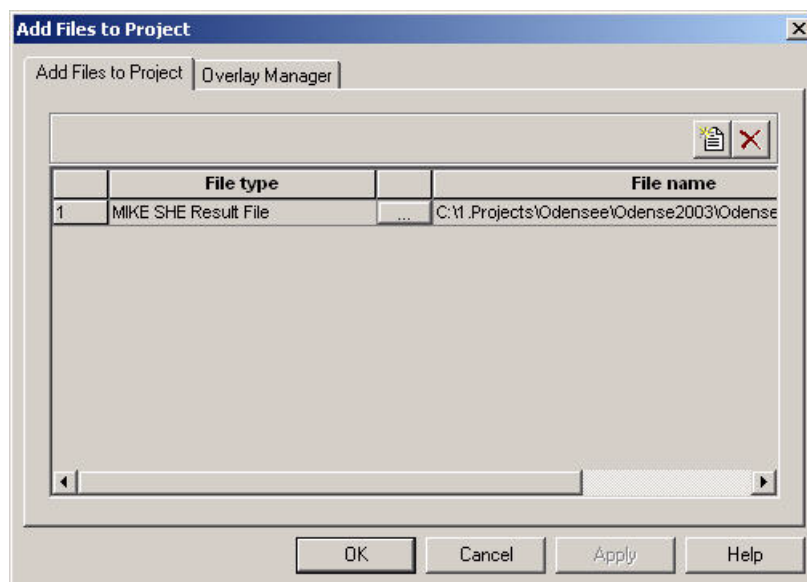
### 7.2.1 Adding additional result files and overlays

A project (or view or plot) in the Results Viewer is a collection of results files and overlays. You can add additional results files or overlays to your current plot by following these steps



- 1 Select Projects/Add Files to Project... from the top pull-down menu. This will open the dialogue below

:

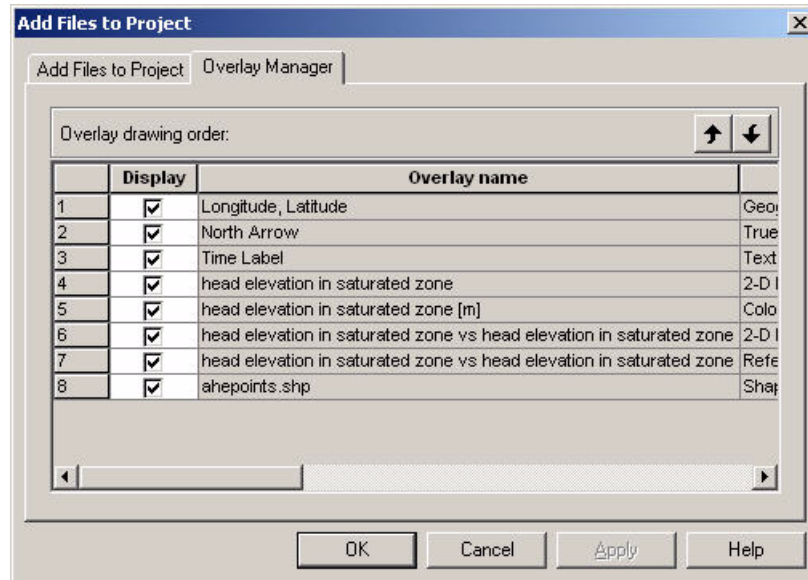


- 2 Click on the Add item button in this dialogue to add a line to the list of files attached to the current project.
- 3 In the left hand column select the type of file to add, including image files, additional results files, and MIKE 11 files.
- 4 Click on the browse button, to find the file that you want to add. All project files will be displayed that are the correct data type.
- 5 If you are adding shape files, you must remember to specify the coordinate axes or the file will not be displayed properly. To do this, you must scroll the dialogue to the right and change the units in the Units combobox.





- 6 After adding the additional file or files, you can modify the drawing order from the Overlay Manager tab

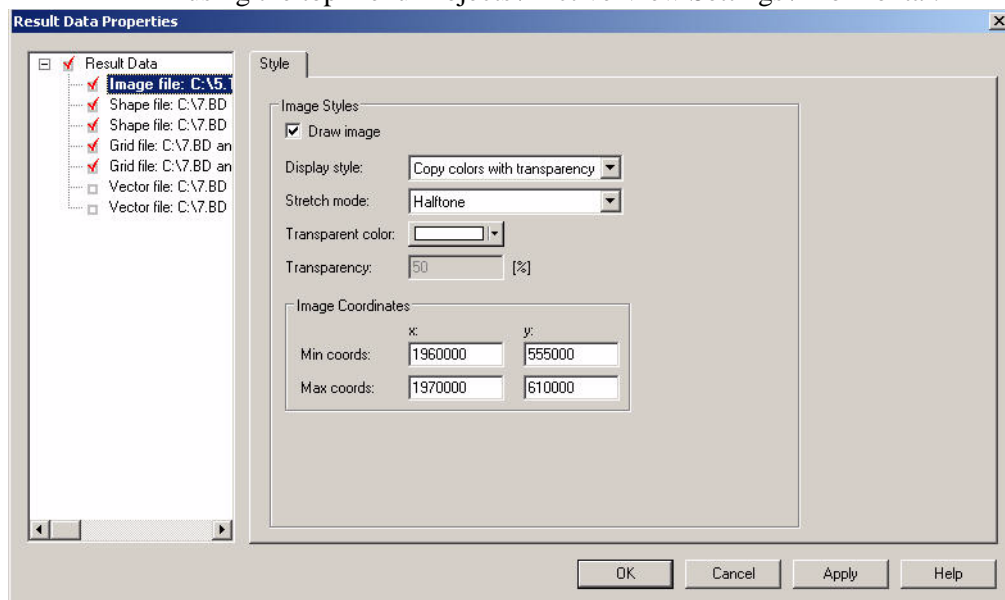


The up arrow and down arrow buttons are used to move an item up or down in the drawing order. The Overlay Manager uses the convention that items are drawn from the lowest to highest item number (i.e., items on the bottom of the overlay list are drawn last and are on top of all other items). The Overlay Manager can also be used to turn overlays on and off by selecting or unselecting overlay items using the check box.

The Overlay Manager can also be accessed from the menu bar by selecting Project / Active View Setting / Overlay Manager.



- 7 After adding the file, open the Property dialogue by right-clicking in the results map and selecting Properties from the pop-up menu or by using the top menu Projects / Active View Settings / Horizontal.



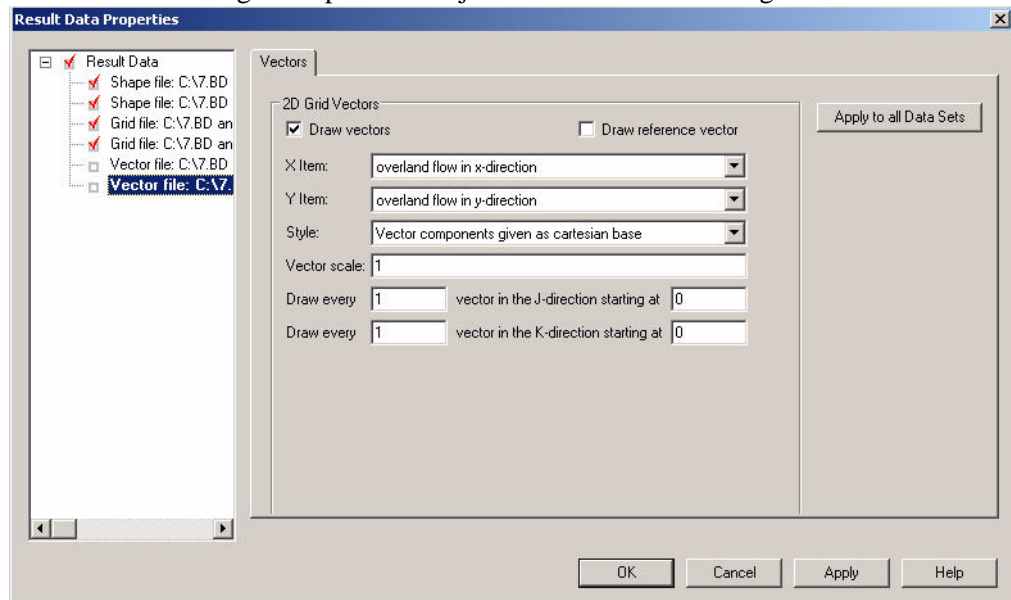
- 8 For an image file (above), you will need to specify the coordinates just like in the Display items in the Setup Tab.

### 7.2.2 Adding or modifying vectors

Vectors can be added by adding a MIKE SHE results files that contains flow data, which are *project\_overland.dfs2* and the *project\_3DSZFlow.dfs3* files. To add vectors follow these steps:



- 1 Add the a flow data file to your results view by following the directions in the section Adding additional result files and overlays (*V.1 p. 95*).
- 2 After adding the flow file, open the Property dialogue by right-clicking in the results map and selecting Properties from the pop-up menu or by using the top menu Projects / Active View Settings / Horizontal.



- 3 When you added the flow results file, the grid data is by default displayed - hiding the original grid data in your results view. To turn off this grid data find the grid item for the data file that you just added and click off the Draw Grid checkbox
  - 4 Then you need to find the Vector item for the flow file that you just added and check on the Draw Vectors checkbox
  - 5 From the comboboxes for X and Y Items, select the flow data for the x and y directions.
  - 6 Finally, select a Vector Scale. A suitable scale can only be found by trial and error. Normally, a large number is good to start with. For example, 10000. If the vectors are too large, then reduce the scale. If they are too small, then increase the scale.
- Note.** There is no vector data for the initial time step in the MIKE SHE results files.
- 7 If your cell size is small or your flows are high you can plot a reduced number of vectors by modifying the Draw every \_\_\_ vector option.



### 7.2.3 Changing the shading or contour settings

The interpolation method can be modified to change the appearance of the Results Viewer display. Available interpolation methods include:

- Box contours without dividers
- Box contours with dividers
- Box contours with transparency
- Shaded contours with copied colours, copy colours with transparency, and blended colours
- No contour

The interpolation method used can be modified by right-clicking in the graphical view and selecting Properties (Figure 7.2) from the pop-up menu or using the Projects / Active View Settings / Horizontal keystrokes and navigating to the grid file entry that you want to modify. The available box contour options are shown in Figure 7.2 and the resulting Result Viewer file using the box contour without and with dividers are shown in Figure 7.1 and Figure 7.3, respectively. The box contour presents simulated results in raster format and uses the exact value generated by the MIKE SHE numerical engine to generate the image shown.

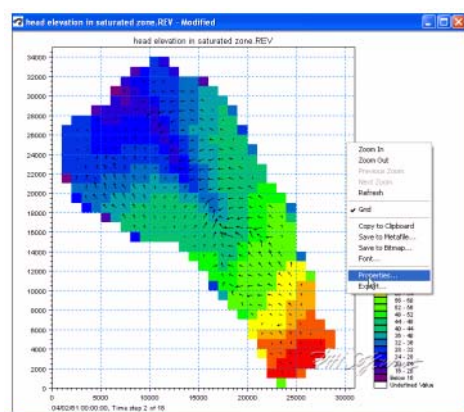


Figure 7.1 Modifying interpolation properties.



Figure 7.2 Specification of box contour options.

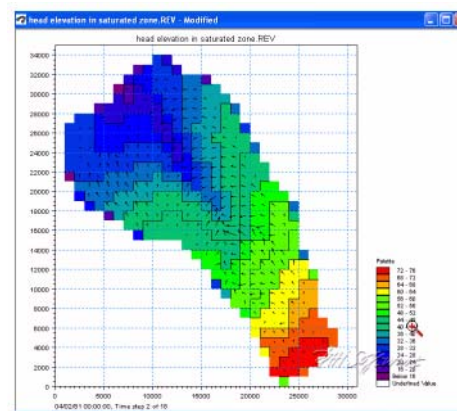
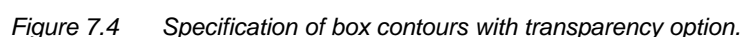
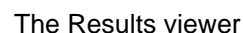


Figure 7.3 Box contours with dividers option.

Available box contour with transparency options are shown in Figure 7.4 and is a useful interpolation option to use to allow underlying overlays to be visible under the plot. The box contour with transparency options presents simulated results in raster format and uses the exact value generated by the MIKE SHE numerical engine to generate the image shown. The amount of transparency can be varied and the resulting plot with 38 % transparency is shown in Figure 7.5.



MIKE SHE



Figure 7.6 Specification of the shaded contour interpolation option with colour blending.

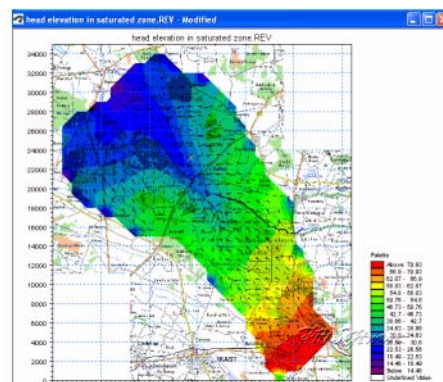


Figure 7.7 Shaded contour interpolation option with colour blending.

Isolines can also be added to all of the interpolation methods to visualize the breaks between contour levels. Isolines can be activated by selecting the isoline check box on the Results Data Properties Style tab for the grid file (Figure 7.8). The resulting Result Viewer file using the blend colour option with isolines is shown in Figure 7.9. As shown on Figure 7.8, addition items can be added to Results Viewer plots including the element mesh (model grid). Users should experiment with various combinations to develop plots that satisfy the intended purpose.



Figure 7.8 Specification of isolines option.

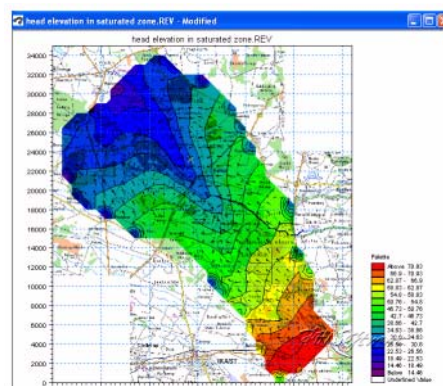


Figure 7.9 Shaded contour interpolation option with colour blending and isolines.

Use of isolines allows plots that display pertinent information without colour contours (no contour option). An example of the Result Viewer plot with isolines but using the no contour option is shown in Figure 7.10.

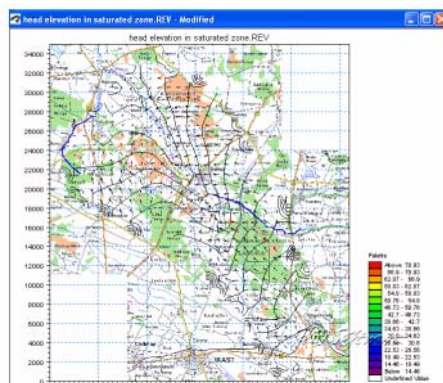


Figure 7.10 Isoline option without contours.





### 7.2.4 Changing the legend and colour scale

In some cases the default colour scheme may not be appropriate for the intended purpose. The colour scheme and/or contour intervals can be modified by right-clicking in the graphical view and selecting Properties (Figure 7.2) from the pop-up menu or using the Projects / Active View Settings / Horizontal keystrokes and navigating to the grid file entry that you want to modify and the Colour tab for the grid entry (Figure 7.11). Options for modifying the colour scheme and/or contour intervals include making a New scheme/contour interval, Editing the existing scheme/contour interval, Opening an existing scheme/contour interval, Saving the current scheme/contour interval, or Resetting (not implemented yet) the current scheme/contour interval to default values. When making a new scheme/contour interval it is possible to modify the Max and/or Min value(s) used to generate the ranges used in the contour intervals (Figure 7.11).

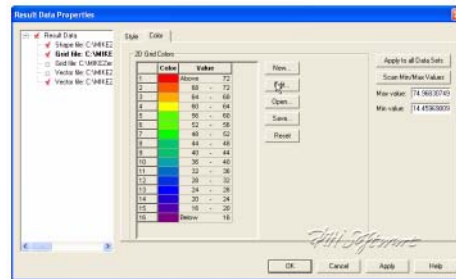


Figure 7.11 Colour modification property tab.

An example of making a new scheme/contour interval using the maximum range is summarized in Figure 7.12 to Figure 7.15. Modification of the number of contour intervals from the default value of 16 to 6 is shown in Figure 7.12. Figure 7.13 shows the available colour schemes that are available and shows use of the Seismic colour scheme. The legend title (Palette title) and palette type (Linear auto Scaled, Fixed, Land/Water Auto Scaled, Land/Water Fixed, and Angle Fixed (Circular)) can also be modified on the first Palette Wizard window (Figure 7.12). Press Next after making the desired changes to move to the next Palette Wizard window.

The colours used for each contour interval (Colour) and the ranges used for each contour interval (Value) can be modified on the second Palette Wizard window (Figure 7.14). Press Next after making the desired changes to move to the next Palette Wizard window.



The third and final Palette Wizard window allows you to review the modified colour scheme and contour intervals before accepting the changes (Figure 7.15). Press the Finish button if all of the modifications are acceptable. Otherwise, press the < Back button to make additional modifications or Cancel button to cancel all changes.



Figure 7.12 Step 1 of 3 - modification of the number of colours used in the colour scheme.



Figure 7.13 Step 1 of 3 - modification of the number of colours model used in the colour scheme.

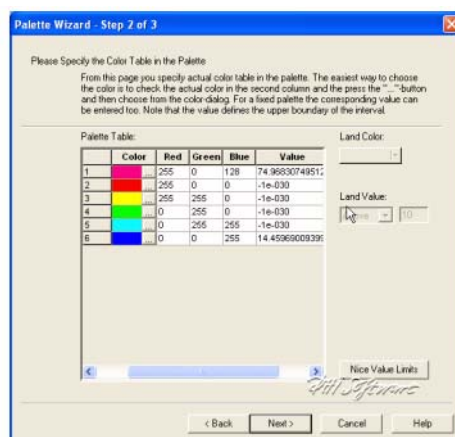


Figure 7.14 Step 2 of 3 - modification of the colours used in the colour scheme and the values colours are applied to.

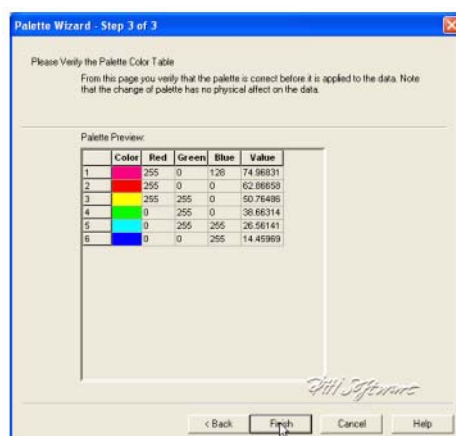


Figure 7.15 Step 3 of 3 - acceptance of the colour scheme modification.

After Accepting the colour scheme/contour interval modifications the Apply button should be pressed on the Result Data Properties window to modify the look of the Result Viewer plot (Figure 7.16). The resulting modified Result Viewer plot is shown in Figure 7.17.

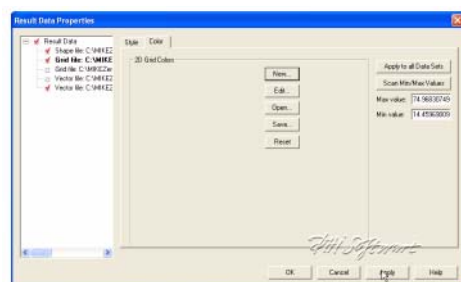


Figure 7.16 Applying the modified colour scheme to the current result viewer file.

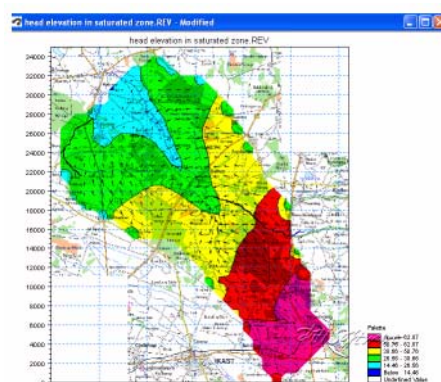


Figure 7.17 Result Viewer file after modification of the default colour scheme.

Users should experiment with the Palette Wizard to develop a better understanding of available functionality than presented in this simple discussion.

### 7.3 Displaying a time series at a point

The Time Series tool allows you to plot a time series of all the data available in the current view.

Time series data can be selected from multiple locations in the active model area using this tool. A single time series can be selected by double-clicking in the desired location. Time series can be extracted from multiple locations by holding down the Ctrl-key and left clicking on each desired location. When selecting multiple locations the Ctrl-key should be held down while double clicking on the last location.



After selecting the locations of the time series files to extract you have the option to deselect some of the selected points and to accumulate the data over the simulation period (Figure 7.18). After making the appropriate selections/deselections press the OK button to generate the time series plot. The entire extraction process can be stopped by pressing the Cancel button. An example of a time series plot generated in the Results Viewer is shown in Figure 7.19.

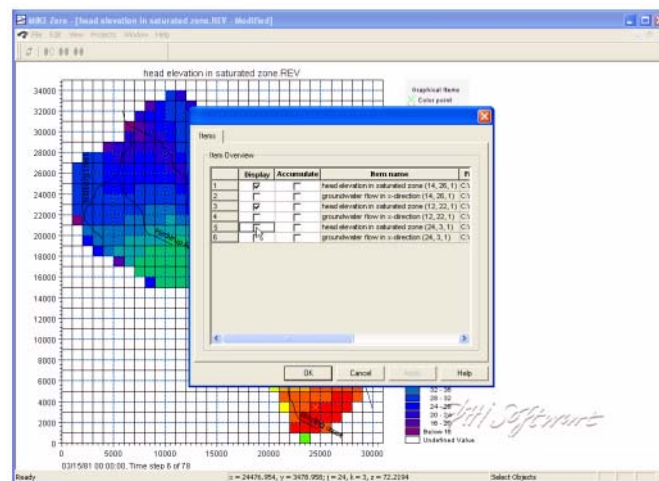


Figure 7.18 Selection of time series items to extract.

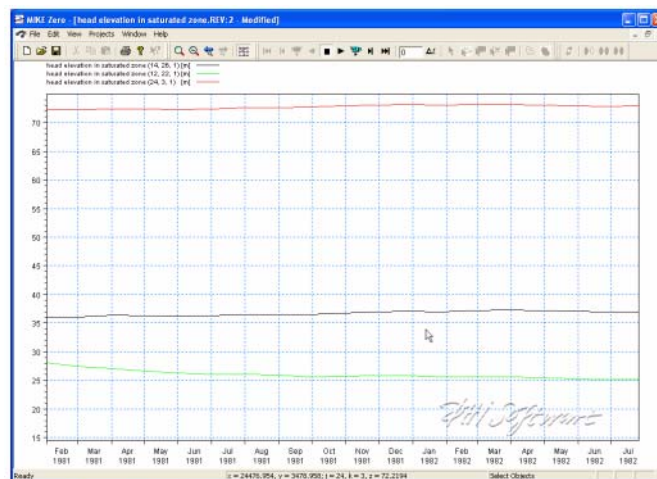


Figure 7.19 Time series plot generated using the time series extraction tool.



Addition graphical functions can be accessed by right-clicking in the graphical view including zooming, exporting images, exporting time series data as dfs0 files, and modification of the time series plot properties (Figure 7.20). Most of the functionality can also be accessed via the menu bar. For example, modification of the time series plot properties can be accessed using Projects/Active View Settings/Timeseries.

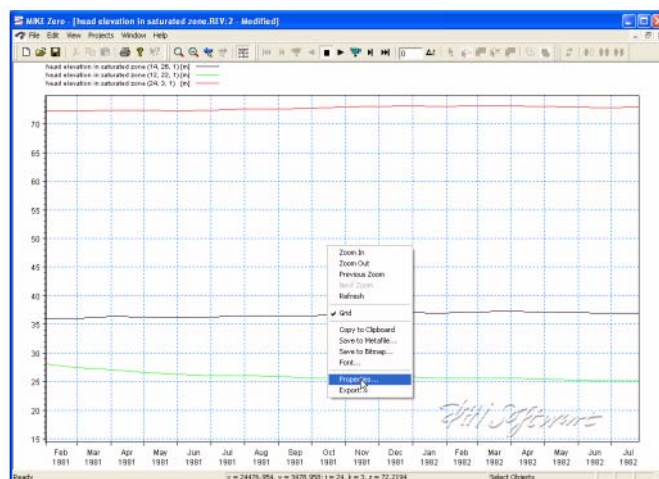


Figure 7.20 Modifying the properties of time series plots in the result viewer.

Because of the rich functionality available in the Result Viewer with respect to time series output, users should experiment with the available options. An example of the available functionality for modification of the time series plot properties is shown in Figure 7.21. For example, as shown in the upper left of Figure 7.21, time series items can be added or deleted from a plot on the items tab.

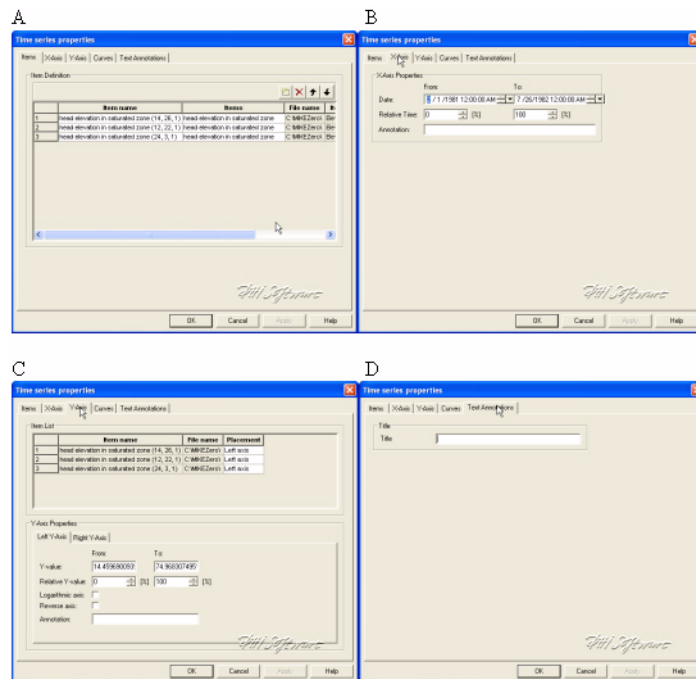


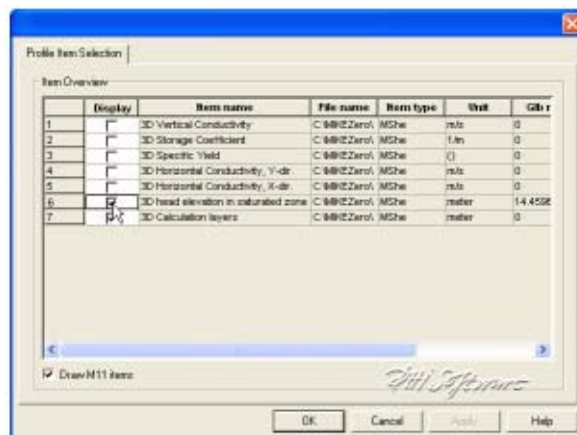
Figure 7.21 Modification of A) items displayed on the time series plot, B) x-axis properties, C) y-axis properties, and D) time series plot title.

## 7.4 Saturated Zone Cross-section Plots

The Profile Extractor tool can be used to extract an arbitrary cross-section through a 3D SZ results file. After you have clicked on this icon, you can define a cross-section profile by left-clicking at each vertex of the profile line and double-clicking to close the profile.



After closing the profile, a dialogue will be displayed that lists the available output types.



Only one of these items can be selected. After selecting your item, click OK and the profile will be displayed.

There are two different types of profiles. The first is a properties profile.

The profiles extractor tool can be used to extract a cross-section through simulated MIKE SHE and MIKE 11 results. The type of cross-section created is dependent on the simulated data displayed in the result viewer. For example, if the result viewer contains simulated 3D heads and MIKE 11 results then the cross-section will have simulated water levels and simulate MIKE 11 canal stages.

After defining the profile, the items to be displayed on the profile should be selected (Figure 7.22). The resulting profile is shown in Figure 7.23. As with the other tools, extracted profiles can be animated on the screen and/or exported as avi and image files.





Figure 7.22 Selection of items to display in a defined profile.

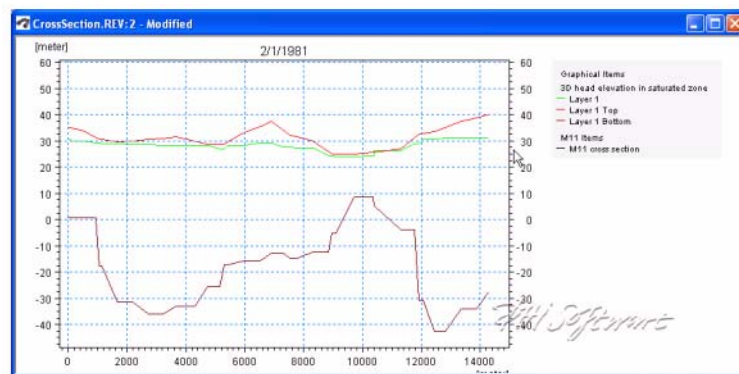


Figure 7.23 Resultant profile generated with the profile extractor tool.

Addition graphical functions can be accessed by right-clicking in the graphical view (e.g. the modification of the profile properties, including changing line and marker properties, adding tabular items, etc.) The available property tabs are shown in Figure 7.24.

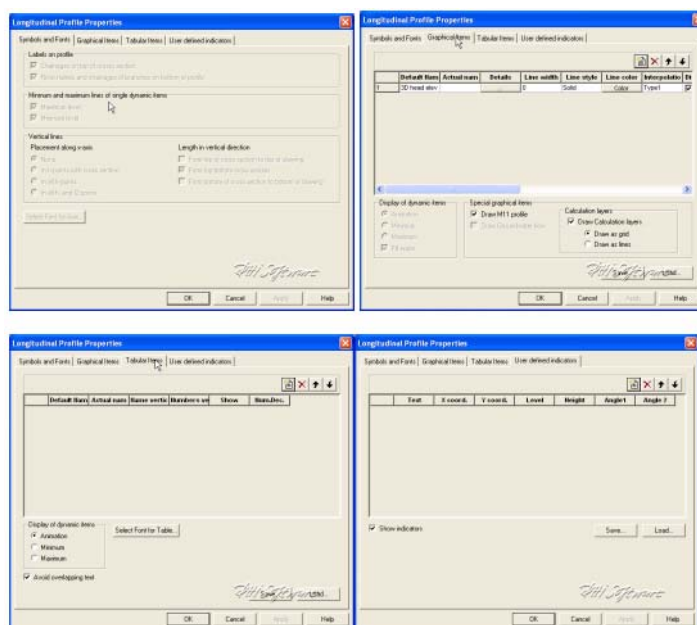


Figure 7.24 Profile property modification windows.

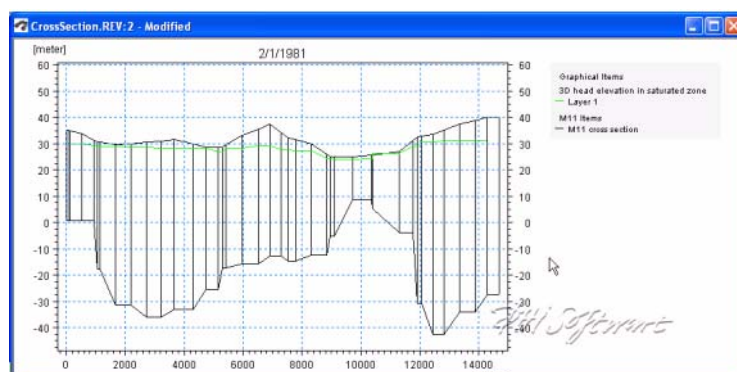


Figure 7.25 Modified profile plot displaying calculation layers as grids instead of lines.

An example of changing the display of the topography and bottom of the calculation layers from lines to grids is shown in Figure 7.25. As with the other tools available in the result viewer, users should experiment with the available options to learn how to fully use the result viewer profile extractor.



#### 7.4.1 Changing to a different SZ layer

### 7.5 Displaying a MIKE 11 cross-section

MIKE 11 results can also be added to the result viewer and simulated canal water levels can be displayed using the cross-section extractor. The cross-section extractor shows simulated stages and the geometry of the cross-section being viewed. The process of adding MIKE 11 results to the result viewer are given in the section Adding additional result files and overlays.

After selecting the cross-section extractor tool, move the cursor over the location you want to extract the MIKE 11 results from (Figure 7.26). The simulated results are displayed along with the cross-section geometry (Figure 7.27). As with the other tools, extracted profiles can be animated on the screen and/or exported as .avi and image files.

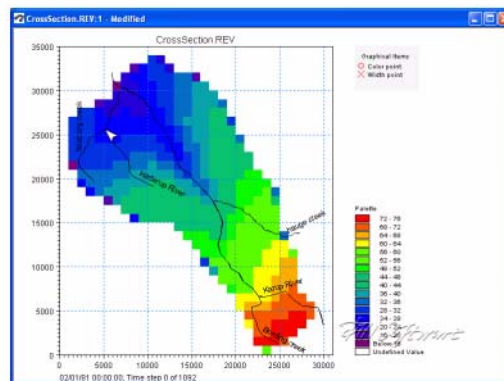


Figure 7.26 Selection of a MIKE 11 cross-section location in the result viewer.

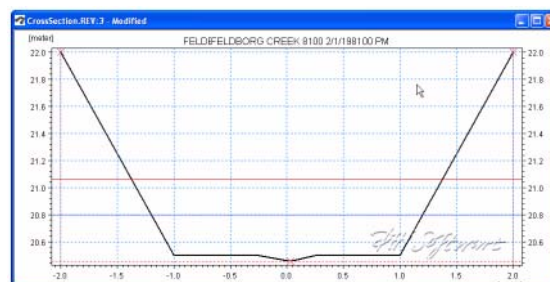


Figure 7.27 Resultant MIKE 11 cross-section plot.



Addition graphical functions can be accessed by right-clicking in the graphical view (Figure 7.28). Modification of the profile properties is one functionality available using the right-click. Since the cross-section plots are relatively simple, modifications are limited to changing line and marker properties, cross-section markers, etc. (Figure 7.29).

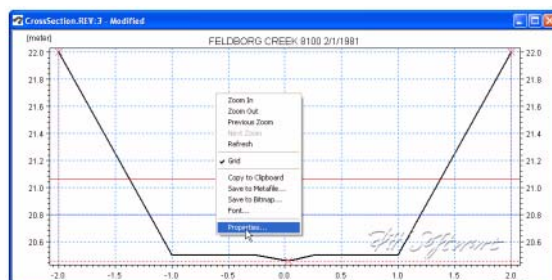


Figure 7.28 Accessing addition functionality in the extracted MIKE 11 cross-section plot.

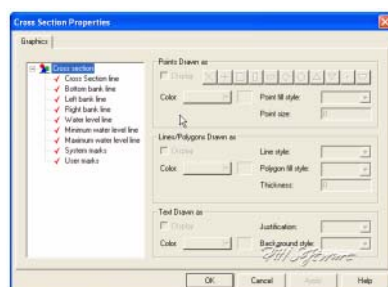


Figure 7.29 MIKE 11 cross-section properties that can be edited.

## 7.6 UZ Specific Plots

### 7.6.1 UZ Scatter and Filled Plots

For unsaturated zone results, scatter or filled plots can be generated. UZ Scatter and Filled Plots are only different for simulations that do not use the “calculation in all cells” UZ module option.

Scatter plots only show simulated results for UZ calculation cells. The number of UZ calculation cells may be less than the total number of active model domain used by the overland and saturated zone modules if the UZ module for the simulation is not using the “calculation in all cells” option. An example of when use of the UZ scatter plot is useful is shown in Figure 7.30.

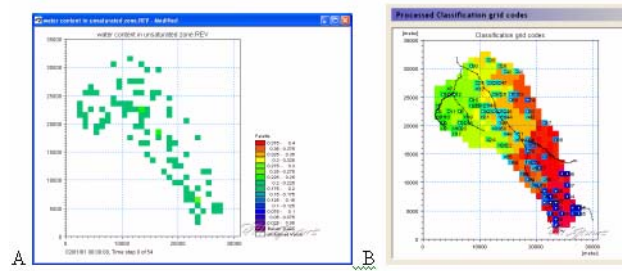


Figure 7.30 A) UZ Scatter Plot and its relationship to B) the UZ calculation cells.

An example of a UZ Filled Plot is shown in Figure 7.31. In cases where the UZ module for the simulation is not using the “calculation in all cells” option the Result Viewer interpolates values from the calculation cells to adjacent inactive UZ cells.

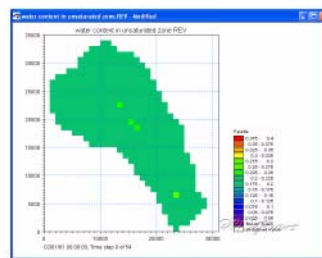


Figure 7.31 Filled UZ plot.

## 7.6.2 UZ Plot

UZ Plots can only be extracted from simulated unsaturated zone water contents and flow. This is because UZ plots display results for a single column for all of the UZ calculation nodes in the column. Other simulated UZ results show net values for the entire UZ (i.e., infiltration, recharge to the SZ, etc.).

After selecting the UZ Plot extractor tool move the cursor over the column you want to extract the results from and double-click (Figure 7.32). Results from multiple UZ columns cannot be displayed on the same UZ Plot.

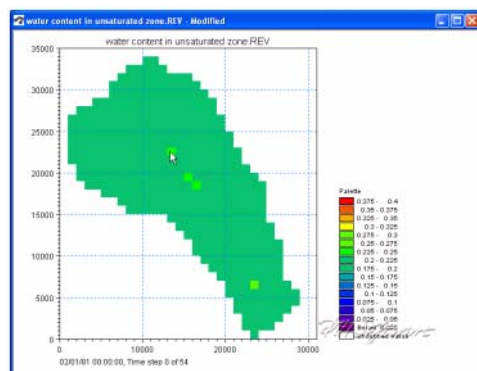


Figure 7.32 Extracting a UZ Plot from simulated unsaturated water contents and flow.

The simulated water content results for the selected column are displayed in Figure 7.33. The UZ Plots show either water content or unsaturated zone flow for each node in the column (y-axis) for the entire simulation (x-axis).

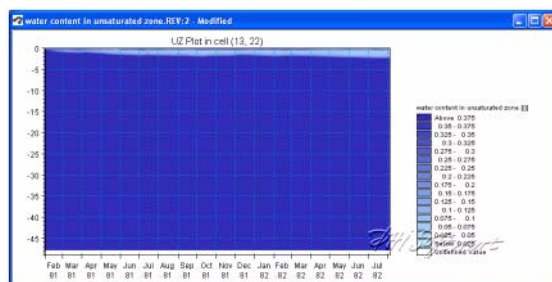


Figure 7.33 Example UZ plot of unsaturated zone water content.

Addition graphical functions can be accessed by right-clicking in the graphical view. Modification of the UZ Plot properties is one functionality available using the right-click (Figure 7.34). Modifications that can be made include changing the interpolation methods, adding the mesh, adding isolines, changing the colour schemes, etc. (Figure 7.35).

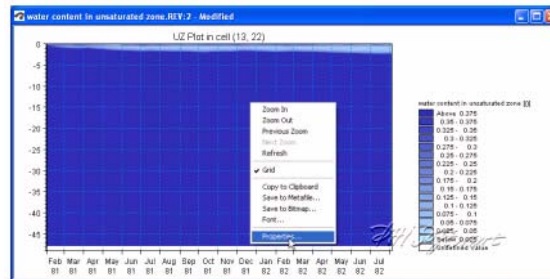


Figure 7.34 Modification of the UZ plot properties.

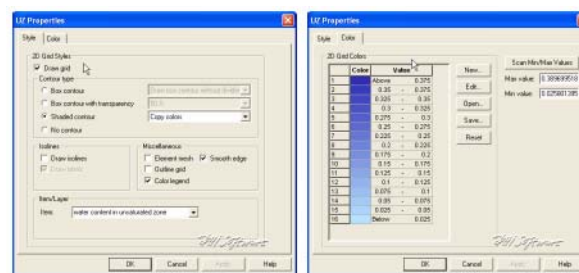


Figure 7.35 Available UZ plot properties that can be modified.

An example of a modified UZ plot with the mesh displayed and only showing the upper five meters of the soil column is shown in Figure 7.35. Additional information on modifying the interpolation and colour scheme are given in the sections Changing the shading or contour settings and Changing the legend and colour scale.

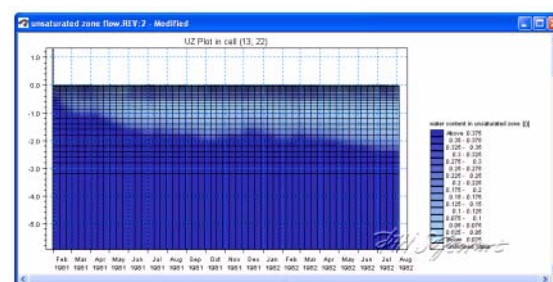


Figure 7.36 Close up of upper 5 meters of soil column with the calculation grid displayed.







## 8 USING THE WATER BALANCE TOOL

The water balance utility is a flexible post-processing tool for generating water balance data for MIKE SHE simulations. Water balance output can include area normalized flows (storage depths), storage changes, and model errors for individual model components (e.g., unsaturated zone, evapotranspiration, etc.).

A water balance can be generated at a variety of spatial and temporal scales and in a number of different formats, including dfs0 time series files, dfs2 grid series files, and ASCII text output suitable for importing to Microsoft Excel. You can also automatically create a picture that visualizes the interrelationships between the various water balance components.

The water balance utility can be run from within the MIKE Zero interface or from a MSDOS batch file. The batch functionality allows you to calculate water balances automatically after a MIKE SHE simulation that is also run in batch mode (see Figure 8.1).

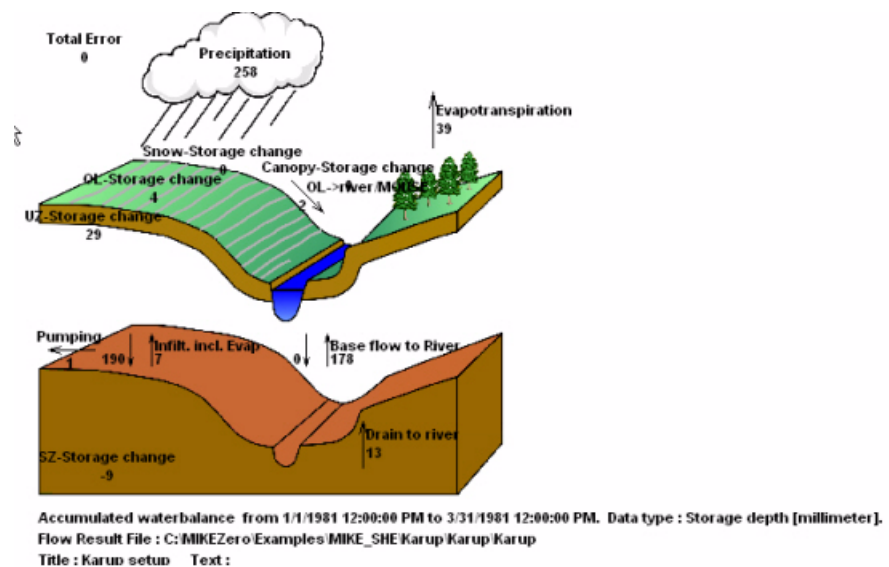


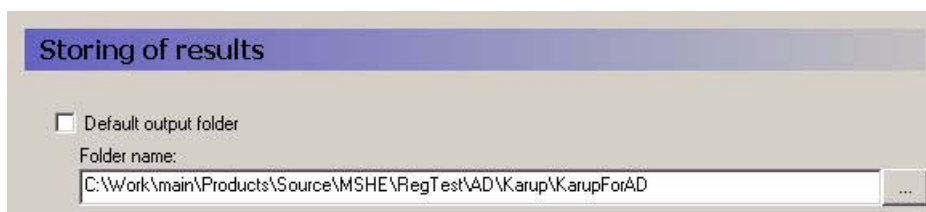
Figure 8.1 Graphical water balance output example

### 8.1 Creating a water balance

Before you can create a water balance for a MIKE SHE WM simulation, you must have saved the water balance data during the simulation. To save or not save the water balance data is specified in the Storing of Results



(V.2 p. 135) dialogue. If you have forgotten to save the water balance data, then you will need to re-run your simulation.

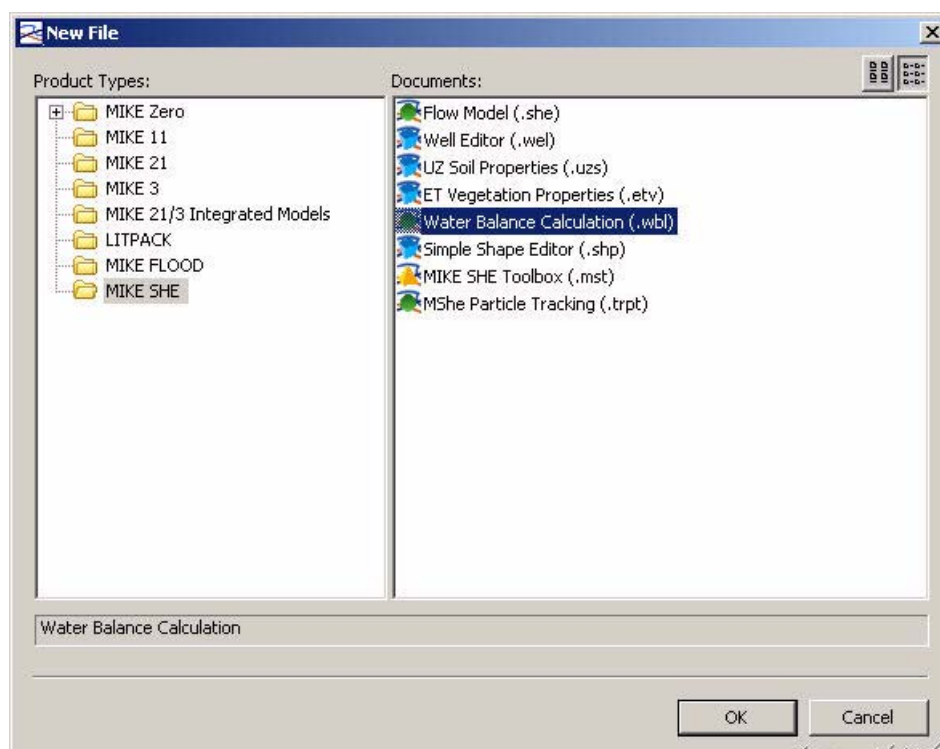


After you have run your WM simulation, creating and running a water balance in MIKE SHE is quite simple, following these steps

- 1 Create a new water balance document (V.1 p. 122),
- 2 Extract the water balance data (V.1 p. 123)
- 3 Specify your water balance (V.1 p. 125), and
- 4 Calculate and View the Water Balance (V.1 p. 129).

#### **8.1.1 Create a new water balance document**

The new water balance document is created by selecting the File/New item in the top menu, or clicking on the New icon in the top menu bar. In the dialogue that appears, select MIKE SHE and Water Balance Calculations in the right hand box, as shown below.



### 8.1.2 Extract the water balance data

To extract the water balance data, you must specify which simulation you are going, then specify the area of your model that you want the water balance for, and, finally, extract the MIKE SHE water balance data from the results files.

Once you have created a new water balance document, the first tab is as shown below.



**Extraction**

Water movement simulation  
Flow result catalogue file: C:\5.Testing\MSHE projects\Qdensee\Qdense2003-hrs ...

Type of extraction  
Area Type: Catchment  
Resolution Type: Area

Sub-catchment grid codes  
Type of input file: Dfs2 Item:   
Dfs2 file: ...

Gross files  
Pre-name of gross files: C:\5\_1\_1 ... ☒ Use default filename

### Flow result catalogue file

A MIKE SHE simulation generates various output files depending on the options and engines selected for the MIKE SHE simulation. The .sheres file is a catalogue of all the various output files generated by the current MIKE SHE run. When you select the .sheres file, you are not specifying the particular output, but actually just a set of pointers to all the output files.

The extraction process reads all of the output files and makes itself ready to produce specific water balances. In the extraction dialogue, you specify the .sheres file for the simulation that you wish to calculate the water balance for. The .sheres file is located in the same directory as your results.

**Note** Although, this is an ASCII file, you should be careful not to make any changes in the file, or you may have to re-run your simulation.

### Type of Extraction

You can choose to calculate the water balance on the entire model domain or in just a part of the domain. By default the calculation is for the entire domain, or catchment. If you choose the subcatchment area type, then you will be able to use a dfs2 integer grid code file to define the areas that you want individual water balances for.



If you use an area resolution, then the water balance will be a summary water balance for either the entire catchment or the sub-areas that you define.

If you use a single-cell resolution, you will be able to generate dfs2 maps of the water balance.

### **Sub-catchment grid codes**


The subcatchment integer grid code file is only used if you have selected the sub-catchment water balance type. You can specify a delete value to exclude areas from the water balance. The grid spacing and dimensions in this dfs2 file must match exactly the model grid.

You can also specify a polygon shape file to define the sub-catchment areas. The shape file may contain multiple polygon, with the same or different codes. Further, the shape file length units do not have to be the same as the model length units (e.g. feet vs. meters).

### **Gross files**

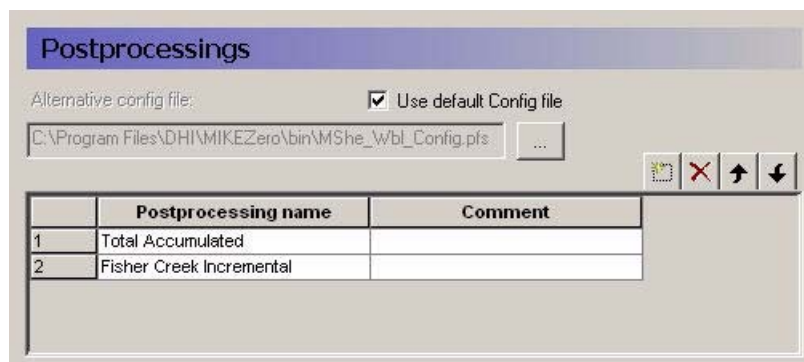
The pre-processor extracts the water balance data from the standard MIKE SHE output files and saves the data in a set of “gross” files. The file names of the gross files is built up from the project name and prefix specified here. The default value is normally fine.

### **Run the extraction**

To run the extraction, you simply have to click on the Run Extraction icon, , or the Run/Extraction top menu item.

#### **8.1.3 Specify your water balance**

After you have extracted the water balance data from the MIKE SHE results files, then you can switch to the post-processing tab. Here you can create any number of individual water balances by simply clicking on the Add item icon and specifying the water balance parameters in the parameter dialogue.



A single Postprocessing item is created by default when the water balance file is created. The default Postprocessing name can be change to a more appropriate name. Postprocessing items that are no longer needed can be deleted using the Delete button.

### Use default Config file

Unchecking the Use default Config file checkbox, allows you to specify the location of a custom water balance Config file. Development of custom water balance configuration files is described in detail in *Making Custom Water Balances (V.1 p. 139)*.

For each item in the Postprocessing list above, a new item will be added to the data tree. If you expand the data tree, each will have the following dialogue.



**Postprocessing**

**Water balance**

Water balance type:

Description:

**Output period**

☒ Use default period    Start date:     End date:

**Output Timeseries Specifications**

☒ Use default output time step    Output time step (hrs):     Type:

**Layer Output Specifications**

Layer:     Layer no.:

**Sub-Catchment Selection**

Grid code:

**Single-Cell Location**

X-index:     Y-index:

**Output File**

Type:     Txt file:  ...

## Water Balance

Multiple postprocessings can be run on each water balance extraction. More detail on the types of available water balances data are discussed in the Available Water Balance Items (*V.1 p. 130*) section. In brief, the available types include

- The total water balance of the entire model catchment or sub-catchments in an ASCII table, a dfs0 file, a dfs2 map file, or a graphical chart (also by layer),
- Model errors for each hydrologic component (overland, unsaturated zone, etc.) in an ASCII table, a dfs0 file, or a dfs2 map file (also by layer),
- The snow melt and canopy/interception water balance in an ASCII table, or a dfs0 file,
- An abbreviated or detailed water balance for overland or unsaturated flow in an ASCII table, or a dfs0 file, and
- An abbreviated or detailed water balance by layer for saturated flow in an ASCII table, or a dfs0 file.



### Output Period

An output period different from the total simulation period can be specified by unchecking **Use default period** and setting the **Start date** and **End date** to the period of interest

### Output Time Series Specification

**Incremental** or **Accumulated** water balances can be calculated. An incremental water balance is calculated (summed) for each output time step in the Output period. An accumulated water balance each output time step is accumulated over the Output period

### Layer Output Specifications

If you are using water balance types that calculate data on a layer basis, you can specify whether you want **All layers** or just the **Specified layer**, where you also must specify a layer number.

### Sub-catchment Selection

If you extracted sub-catchment data from the WM results, then you must specify a subcatchment number or the name of the polygon for which you want the water balance for. The combobox contains a list of valid ID numbers or polygon names.



### Single Cell Location

If you extracted the WM data by cell, then if you are not creating a map output, then you have to specify a cell location for which you want a water balance.

### Output File

If you are creating a table or time series water balance, then you can write the output to either a dfs0 file or to an ASCII file for import to MSExcel, or other post-processing tool. If you are creating a map, then the output will be to a dfs2 file, with the same grid dimensions and spacing as the model grid. If you are creating a chart, then the output will be written to an ASCII file, with a special format for creating the graphic.

### Run the Post Processing

To run the post processing, you have two choices. You can click on the Run Selected Post-Processing icon, , which runs only the current post-processing item. Or, you can click on the Run All Post-Processing icon, , which runs all of the post-processing items in the list. These two options are also available in the Run top menu.





#### 8.1.4 Calculate and View the Water Balance

The data tree for the results tab lists all of the calculated water balances. The dialogue for each item, includes the file name and an Open button, that will open an editor for the file. For ASCII output, this will be your default ASCII editor - usually Notepad. For dfs0 and dfs2 files, the DHI Time Series Editor or Grid Editor will be opened. For the chart output, the graphic will be displayed by the program WblChart.

## 8.2 Calculating Water Balances in Batch Mode

Like most DHI software, the water balance utility can be run in batch mode. Some possible ways to run the water balance utility in batch mode are:

- Running the water balance utility immediately after completion of a MIKE SHE simulation run in batch mode.
- Running the water balance utility for a MIKE SHE simulation without using the water balance utility graphical users interface.
- Running multiple water balance Postprocessing stages automatically without using the water balance utility graphical users interface.

The water balance graphical utility stores all of its information in a .wbl file. The .wbl file is an ASCII file that can be edited with Notepad or other text editor, but the format of the water balance file must be preserved. You can open and review several .wbl files created by the water balance utility to get an understanding of the file format and structure.

**Note**, we cannot provide support to debug configuration files created outside of the user interface.

To run the water balance utility in batch mode, the .wbl file must be created prior to executing it and all file names in the .wbl file need to be valid. If during calibration the same MIKE SHE file name is used for each simulation then the same water balance file can be used for all calibration runs. If the MIKE SHE simulation to be evaluated is different from the MIKE SHE simulation used to set up the water balance file, you will have to edit the water balance file.

To run the Extraction and Postprocessing steps in batch mode, the your PATH statement needs to include directory where MIKE SHE was installed. The default directory is

C:\Program Files\DHI\MIKEZero\bin



An example is shown below of a batch file that generates water balance data for three postprocessing steps, using a water balance utility file named *WaterConservationAreas.WBL*.

```
rem -----  
MSHE_Wbl_Ex.exe WaterConservationAreas.WBL  
MSHE_Wbl_Post.exe WaterConservationAreas.WBL 1  
MSHE_Wbl_Post.exe WaterConservationAreas.WBL 2  
MSHE_Wbl_Post.exe WaterConservationAreas.WBL 3
```

The `MSHE_Wbl_Ex.exe` command runs the Extraction phase of the water balance utility. The command

```
MSHE_Wbl_Post.exe WaterConservationAreas.WBL 1
```

runs the first Postprocessing item in the water balance file, *WaterConservationAreas.WBL*. The number after the water balance file name in the Postprocessing command indicates which Postprocessing item to run and this number must be consistent with the water balance utility file (i.e., the number cannot be greater than the number of Postprocessing items in the file). Otherwise, the program will terminate with an error. The Postprocessing step cannot be executed before an Extraction step but only one Extraction step needs to be run for a single water balance utility file.

The water balance batch file can contain Extraction and Postprocessing steps from multiple water balance utility files.

### 8.3 Available Water Balance Items

The water balance utility is a very flexible tool that allows you to modify existing Water balance types or create custom Water balance types to suit your needs. The water balance calculations use a water balance Configuration file to define Water balance types using the available water balance items and a macro language to control program execution.

To modify existing or custom Water balance types you must understand the available items and what data they contain.

**Note** The water balance utility and MIKE SHE uses a positive upward coordinate system. In other words, water flowing upward is a positive flow in the water balance.

The following tables outline the various water balance items:



- Table 8.1 SM - Precipitation and snowmelt items (*p. 131*)
- Table 8.2 CI - Canopy interception items (*p. 131*)
- Table 8.3 OL - Overland flow items (*p. 132*)
- Table 8.4 UZ - Unsaturated Zone items (*p. 133*)
- Table 8.5 SZ - Saturated Zone items (*p. 134*)

*Table 8.1 SM - Precipitation and snowmelt items*

| Item             | Description                              | Sign   |
|------------------|--|--|
| sm.qp            | Precipitation                            | Positive upwards (precipitation is negative and dew is negative) |
| sm.qpad          | Precipitation minus snow and evaporation | Positive upwards (always negative)                               |
| sm.dsnowsto      | Change in snow storage                   | Positive when snow storage increases                             |
| sm.qesnow        | Evaporation from snow                    | Positive out (always negative)                                   |
| sm.qirrsprinkler | Sprinkler Irrigation                     | Positive out (always negative)                                   |
| sm.smwblerr      | SM Water balance error                   | Positive if the sum of SM inflows exceeds SM outflows            |

*Table 8.2 CI - Canopy interception items*

| Item        | Description  | Sign  |
|-------------|--|---|
| ci.qpad     | Precipitation reaching Canopy - Precipitation minus snow | Positive upwards (always negative)                    |
| ci.qpnet    | Canopy through fall                                      | Positive upwards (always negative)                    |
| ci.qeint    | Evaporation from intercepted water                       | Positive out (always positive)                        |
| ci.dintsto  | Change in interception storage                           | Positive when storage increases                       |
| ci.ciwblerr | CI Water balance error                                   | Positive if the sum of CI inflows exceeds CI outflows |



Table 8.3 OL - Overland flow items

| Item             | Description   | Sign                                   |
|------------------|---|--|
| ol.qpnet         | Canopy through-fall   | Positive up (always negative)          |
| ol.qh            | Infiltration from OL to UZ  | Positive to OL (always negative)       |
| ol.qhmp          | Infiltration from OL to UZ macropores                                     | Positive to OL (always negative)       |
| ol.qeol          | Evaporation from ponded water   | Positive out (always positive)         |
| ol.qolin         | Inflow to Overland from boundary of sub-catchment                         | Positive to boundary (always negative) |
| ol.qolout        | Outflow from Overland to boundary of sub-catchment                        | Positive to boundary (always positive) |
| ol.qolrivpos     | Overland outflow to MIKE 11 river   | Positive to MIKE 11 (always positive)  |
| ol.qocdr         | Overland flow (from paved areas) directly to MIKE 11                      | Positive to MIKE 11 (always positive)  |
| ol.qolszpos      | Upward flow from SZ to OL   | Positive to OL (always positive)       |
| ol.qolszneg      | Downward flow from OL to SZ   | Positive to OL (always negative)       |
| ol.qsztofloodpos | Outflow from SZ to flooded areas  | Positive to OL (always positive)       |
| ol.qsztofloodneg | Inflow to SZ from flooded areas   | Positive to OL (always negative)       |
| ol.qfloodtorivin | Exchange from overland flooded areas to MIKE 11 inside the sub-catchment  | Positive to MIKE 11                    |
| ol.qfloodtorivex | Exchange from overland flooded areas to MIKE 11 outside the sub-catchment | Positive to MIKE 11                    |



Table 8.3 OL - Overland flow items

| Item           | Description                    | Sign   |
|----------------|--------------------------------|--|
| ol.dolsto      | Change in overland storage     | Positive when storage increases                        |
| ol.qirrdrip    | Drip Irrigation                | Negative to OL (always negative)                       |
| ol.qolMousepos | Inflow to Overland from Mouse  | Positive to OL (always positive)                       |
| ol.qolMouseneg | Outflow from Overland to Mouse | Positive to OL (always positive)                       |
| ol.olwblerr    | OL water balance error         | Positive if the sum of OL inflows exceeds OL out-flows |

Table 8.4 UZ - Unsaturated Zone items

| Item             | Description  | Sign                          |
|------------------|--|-------------------------------|
| uz.qh            | Infiltration from Overland to Unsaturated Zone                   | Positive in (always positive) |
| uz.qhmp          | Infiltration from Overland to Unsaturated Zone Macro-pores       | Positive in (always positive) |
| uz.qrech         | Infiltration from Unsaturated Zone to Saturated Zone             | Positive to UZ                |
| euz.qrechmp      | Infiltration from Unsaturated Zone Macro-pores to Saturated Zone | Positive to UZ                |
| uz.qtuz          | Transpiration from the root zone                                 | Positive up (always positive) |
| uz.qeuz          | Direct evaporation from soil                                     | Positive up (always positive) |
| uz.qgwfeedbackuz | Feedback from LR to UZ   | Positive from LR              |
| uz.duzdef        | Change in UZ deficit   | Positive when increasing      |



Table 8.4 UZ - Unsaturated Zone items

| Item            | Description   | Sign  |
|-----------------|---|---|
| uz.uzszstorcorr | Correction to account for difference in unconfined storage capacity for UZ and SZ | Positive when storage increases                       |
| uz.uzwblerr     | UZ Water balance error  | Positive if the sum of UZ inflows exceeds UZ outflows |

Table 8.5 SZ - Saturated Zone items

| Item                     | Description  | Sign  |
|--------------------------|--|---|
| sz.qrech                 | Recharge from UZ to SZ   | Positive to UZ  |
| sz.qrechmp               | Recharge from UZ macropores to SZ  | Positive to UZ  |
| sz.qetsz                 | Evapotranspiration directly from SZ  | Positive up (always positive)                         |
| sz.qolszpos              | Upward flow from SZ to overland  | Positive up (always positive)                         |
| sz.qolszneg              | Downward flow from overland to SZ  | Positive up (always negative)                         |
| sz.szwblerrtot           | SZ water balance error for sub-catchment   | Positive if the sum of SZ inflows exceeds SZ outflows |
| sz.qszprecip (per layer) | Precipitation added directly to SZ layer   | Positive up   |
| sz.qszout (per layer)    | SZ flow out of the subcatchment  | Positive out of SZ (always positive)                  |
| sz.qszzpos (per layer)   | Upward SZ flow from the current layer (layer i) to the layer above (layer i-1)<br>only available for LAYER DETAIL type | Positive up   |



Table 8.5 SZ - Saturated Zone items

| Item                           | Description   | Sign                                    |
|--------------------------------|---|---|
| sz.qszzneg<br>(per layer)      | Downward SZ flow from the layer above (layer i-1) to the current layer (layer i)<br>only available for LAYER<br>DETAIL type | Positive up<br>(always negative)        |
| sz.dszsto<br>(per layer)       | Change in SZ storage  | Positive when storage<br>increases      |
| sz.qszabsex<br>(per layer)     | Groundwater (SZ) pumping  | Positive up                             |
| sz.qszdrin<br>(per layer)      | SZ drainage into subcatchment   | Positive out of SZ (always<br>negative) |
| sz.qszdrout<br>(per layer)     | SZ drainage out of subcatch-<br>ment  | Positive out of SZ (always<br>positive) |
| sz.qszdrtorivin<br>(per layer) | SZ drainage flow to MIKE 11<br>reaches inside the subcatch-<br>ment   | Positive out (always posi-<br>tive)     |
| sz.qszdrtorivex<br>(per layer) | SZ drainage flow to MIKE 11<br>reaches outside the subcatch-<br>ment  | Positive out (always posi-<br>tive)     |
| sz.qszdrtoMouse<br>(per layer) | SZ drainage to MOUSE man-<br>holes  | Positive out of SZ                      |
| sz.qszrivpos<br>(per layer)    | SZ baseflow to MIKE 11  | Positive out (always posi-<br>tive)     |
| sz.qszrivneg<br>(per layer)    | MIKE 11 baseflow to SZ  | Positive out (always nega-<br>tive)     |
| sz.qgihbpos<br>(per layer)     | SZ flow to internal head<br>boundaries (GHB cells)  | Positive out (always posi-<br>tive)     |



Table 8.5 SZ - Saturated Zone items

| Item                          | Description   | Sign  |
|-------------------------------|---|---|
| sz.qgihbneg<br>(per layer)    | Flow from internal head boundaries (GHB cells) to SZ                          | Positive out (always negative)  |
| sz.qszMousepos<br>(per layer) | SZ flow to Mouse  | Positive out of SZ (always positive)  |
| sz.qszMouseneg<br>(per layer) | Mouse flow to SZ  | Positive out of SZ (always negative)  |
| sz.qirrwell<br>(per layer)    | Pumping from SZ for irrigation  | Positive out of SZ (always positive)  |
| sz.szwblerr<br>(per layer)    | SZ water balance error for layer i<br>only available for LAYER<br>DETAIL type | Positive if the sum of SZ inflows for layer i exceeds SZ outflows for layer i |

## 8.4 Standard Water Balance Types

The first combobox in the Post-processing dialogue contains a list of all the available water balance types. This list is read from the water balance configuration file, `MSHE_Wbl_Config.pfs`, which is found in the MIKE SHE installation bin directory. By default, this directory is

`C:\Program files\DHI\MIKEZero\bin\`

Table 8.6 summarizes the 31 standard water balance types defined in the water balance configuration file. Some of the water balances cannot be used in certain conditions and these restrictions are listed in the table.

Table 8.6 Water balance types available in the default configuration files.

| Water balance type      | Description                                     |
|-------------------------|---|
| Total waterbalance      | General water balance of the entire model setup |
| Error of each component | The water balance error of each model component |





Table 8.6 Water balance types available in the default configuration files.

| Water balance type                              | Description   |
|---|---|
| Snow Melt component                             | Snow Melt component water balance items                                   |
| Canopy Interception component                   | Canopy Interception component water balance items                         |
| Overland flow                                   | Overland component water balance  |
| Overland flow - detailed                        | Detailed Overland component water balance                                 |
| Unsaturated Zone                                | Unsaturated zone component water balance                                  |
| Unsaturated Zone - detailed                     | Unsaturated zone component water balance                                  |
| Saturated Zone                                  | Saturated zone component water balance (depth-integrated)                 |
| Saturated Zone - layer(s)                       | Saturated zone component water balance (each or specified layer)          |
| Saturated Zone - detailed                       | Detailed Saturated zone component water balance (depth-integrated)        |
| Saturated Zone - detailed - layer(s)            | Detailed Saturated zone component water balance (each or specified layer) |
| Saturated Zone (Linear Reservoir)               | Saturated Zone component water balance for the linear reservoir           |
| Saturated Zone -layers (Linear Reservoir)       | Saturated Zone component water balance for the linear reservoir           |
| Irrigation component                            | Irrigation component water balance  |
| MOUSE-coupling terms                            | MIKE SHE - MOUSE exchange (depth-integrated)                              |
| MOUSE-coupling terms, Saturated zone - layer(s) | MIKE SHE sat.zone - MOUSE exchange (each or specified layer)              |
| Map output: Total error                         | Distributed output: Total water balance error                             |
| Map output: Overland flow error                 | Distributed output: Overland water balance error                          |
| Map output: Unsat. Zone error                   | Distributed output: Unsat.zone water balance error                        |
| Map output: Sat. Zone error                     | Distributed output: Saturated zone water balance error (depth-integrated) |



Table 8.6 Water balance types available in the default configuration files.

| Water balance type                                 | Description  |
|--|--|
| Map output: Sat. Zone error - layer(s)             | Distributed output: Saturated zone water balance error (each or specified layer) |
| Map output: Total irrigation                       | Distributed output: Total irrigation   |
| Chart output: Total water balance                  | Chart output: General water balance of the entire model (depth-integrated)       |
| Chart output: Total + each SZ layer                | Chart output: General water balance of the entire model (each SZ layer)          |
| Chart output: Total water balance TEXT IN DANISH   | Chart output: Generel vandbalance for hele modellen (dybde-integreret)           |
| Chart output: Total + each SZ layer TEXT IN DANISH | Chart output: Generel vandbalance for hele modellen (hvert SZ-lag)               |
| Saturated Zone                                     | StorageSaturated zone Storage (depth-integrated)                                 |
| Saturated Zone Storage - layer(s)                  | Saturated zone Storage (each or specified layer)                                 |
| Map output: Saturated Zone Storage                 | Distributed output: Saturated zone Storage (depth-integrated)                    |
| Map output: Saturated Zone Storage - layer(s)      | Distributed output: Saturated zone Storage (each or specified layer)             |

Below is an extract from the water balance configuration file to illustrate how a water balance is assembled.

```
[WblTypeDefinition]
Name = 'UZ'
DisplayName = 'Unsaturated Zone'
Description = 'Unsaturated zone component water balance'
NoGroups = 6
Group = 'Infiltration(uz.qh+uz.qhmp) '
Group = 'Evaporation(uz.qeuz) '
Group = 'Transpiration(uz.qtuz) '
Group = 'Recharge(-uz.qrech-uz.qrechmp-uz.qgwfeedbackuz) '
Group = 'Deficit Change(-uz.duzdef+uz.uzszstocorr) '
Group = 'Error(uz.uzwblerr) '
EndSect // WblTypeDefinition
```



## 8.5 Making Custom Water Balances

All of the water balances in the Water Balance Type combobox of the Postprocessing dialogue are defined in the water balance configuration file. You can add extra items to this list by defining additional water balances at the end of the configuration file.

To illustrate how you could add an additional water balance type, the table below describes the format for each line of the water balance type definition. The example is for an extra water balance type to calculate the net vertical flow in a specified SZ layer. This water balance type can only be used with the single-cell resolution and specified output layers options.

| Line item   | Comment   |
|---|---|
| // Created: 2004-06-2 16:28:48<br>// DLL id : C:\WINDOWS\System32\pfs2000.dll<br>// PFS version: Mar 3 2004 21:35:12<br><br>[MIKESHE_WaterBalance_ConfigFile]<br>FileVersion = 3<br>NoWblTypes = 31 | File header<br><br>NoWblTypes = the number of water balance types in the configuration file. Remember to change this number if you add a water balance item to the file                             |
| [WblTypeDefinition]<br>Name = 'TOTAL'<br>...<br>Group = 'SZ Storage(sz.szsto)'<br>EndSect // WblTypeDefinition  | Existing water balance definitions  |
| [WblTypeDefinition]   | First line of the water balance definition  |
| Name = 'SZ_LAYER_NET_VERT_FLOW_MAP'   | Internal name. No spaces allowed  |
| DisplayName = 'Map output: Net Vertical Saturated Zone Flow - layer(s)'   | Name displayed in the combobox  |
| Description = 'Distributed output: Saturated zone Storage (specified layer)'  | Description displayed under the combobox  |
| NoGroups = 1  | Number of calculation groups in the output file   |
| Group = 'SZ Vertical Flow(sz.qszzpos+sz.qszzneg)'   | Definition of the calculation group, consisting of a name and a sum of the particular water balance items (no spaces) from Table 8.1 to Table 8.5. Map items can only have one group (NoGroups = 1) |
| EndSect // WblTypeDefinition  |   |
| EndSect // MIKESHE_WaterBalance_ConfigFile  | last line in the file   |



When making custom water balance types the format of the default water balance configuration file must be maintained. Variable names, including names in square brackets, are case sensitive and the number of spaces in variable names must be consistent with the default configuration file. Unfortunately, we cannot provide for support for custom water balance files.

## **8.6 Water Balance Restrictions**

### **8.6.1 Linear Reservoir and Simple Sub-catchment Overland Flow**

The water balance calculations have the following restrictions on **single-cell, sub-catchment** water balances, with the SZ Linear Reservoir and Simple OL:

- single-cell : won't be correct for TOTAL, OL, SZ water balances. But can be used for UZ and others.
- sub-catchment: For TOTAL and OL water balances the smallest valid water balance sub-catchment is one Overland flow zone (i.e. topographical zone) within one hydrological sub-catchment. If a water balance sub-catchment excludes part of an Overland flow zone within one hydrological sub-catchment, the water balances will be wrong in many cases because the OL storage is not necessarily uniformly distributed over one Overland flow zone, while there is only one value for flows between OL flow zones, source/sink terms, etc.
- For TOTAL and SZ water balances: Same restrictions apply, but here with the interflow reservoirs.

There are no restrictions with respect of the baseflow reservoir distributions.

The pre-processor warns in case the above restrictions are violated. It can't give an error, because this program doesn't know which type (TOTAL/OL/SZ/...) the user will specify in the water balance Post-processor.



*Phi Software*

## **ADDITIONAL OPTIONS**





## 9 **EXTRA PARAMETERS**

The Extra Parameters section is a special section of the Setup data tree that allows you to input parameters for options that have not yet been included in the MIKE SHE user interface.

The Extra Parameters are only recognized if the Name (e.g. “sheet piling module”) are spelled exactly correct. After the initial run, you should check in the Preprocessor\_print.log file to ensure that the module has actually been activated.

Available Extra Parameters include:

- Sheet Pile Module (*V.1 p. 143*)
- Negative Precipitation (*V.1 p. 148*)
- Precipitation Multiplier (*V.1 p. 149*)
- SZ Drainage to Specified MIKE 11 H-points (*V.1 p. 149*)
- Time varying drainage parameters (*V.1 p. 152*)
- Distributed Snow Melt Constants (*V.1 p. 155*)
- Canyon exchange option for deep narrow channels (*V.1 p. 155*)
- Simplified Overland Flow Options (*V.1 p. 156*)
- 2-Layer UZ Options (*V.1 p. 157*)
- GeoViewer Output (*V.1 p. 158*)

### 9.1 **Sheet Pile Module**

The Sheet Piling module is not yet included in the MIKE SHE GUI. However, the input for the module is fairly simple and is handled via the Extra Parameters options



The Sheet Piling module is activated by including the following two parameters in the Extra Parameters section of the data tree, and creating the required module input file:

| Parameter Name      | Type      | Value                                      |
|---------------------|-----------|--|
| sheet piling module | Boolean   | On   |
| sheet piling file   | file name | the file name of the Sheet Pile input file |

### 9.1.1 Sheet Pile Location

The location of the sheet piles is defined using a dfs2 file with integer grid codes. One file (or item) is required for each computational layer with sheet piling. Each file must have the same grid size as the MIKE SHE model. The grid codes are “composed” of simple sums of 100, 10, 1, 0 where:

100 = a N-S sheet piling “link” between the actual cell and the next cell in positive x-direction,

10 = a E-W sheet piling “link” between the actual cell and the next cell in the positive y-direction,

1 = a Horizontal sheet-piling “surface” between the actual layer and the layer above (ground surface if actual layer is 1), and

0 = no sheet piling.

Thus, for example, a cell containing the code “110” defines the existence of sheet piling along the Eastern and Northern cell boundaries. A cell containing the code “11” defines a sheet piling along the Northern cell boundary and at the top of the layer.

### 9.1.2 Leakage Coefficient

The Leakage Coefficient is required for flow in the x-, y-, and z-direction for each layer containing sheet piling. The Leakage Coefficient is required in the x-direction if any cell contains a “100” value, in the y-direction if any cell contains a “10” value, and in the z-direction if any cell contains a “1” value.

The leakage coefficients can be specified as a global value (per layer) or as a distribution in a dfs2 file. In the case of a dfs2 file, the values must be specified in the cells where the grid codes are specified. The EUM type (unit) of the dfs2 files must be “Leakage coefficient/Drain time constant” with the unit 1/Time.





### 9.1.3 Top and bottom levels (optional)

This option can be used when the vertical sheet piling only extends across part of a layer. The levels are specified in the same cells as the leakage coefficients in the x- and y-direction, one set of top and bottom levels for each direction.

The levels can be specified as global values (per layer) or as a distribution in a dfs2 file. Both can be absolute levels or relative to ground. The EUM type of the dfs2 files must be “elevation” for absolute levels, and “depth below ground” (positive values) or “height above ground” (negative values) when specified relative to the ground surface. The type and unit of the global value is “elevation” (m) when absolute, and “height above ground” (m) (negative value) when relative.

In cells where the sheet pile extends across the entire layer, the top and bottom levels should simply be set to large positive and negative values respectively (e.g. 1.0E+30 and -1.0E+30).

### 9.1.4 Input File for the Sheet Pile Module

The name of the input file is specified in the Extra Parameters section described above. The file has the general MIKEZero parameter file (pfs) format. The exact format of the file is given below, along with a description of the different data items:

| Line item  | Comment   |
|--|---|
| [MIKESHE_SheetPiling_File]<br>FileVersion = 2<br>[SheetPiling] | FileVersion can be 1 or 2, but must be 2, if you want to check for the SpecifiedXYLevels option                           |
| NrOfLayers = 1   | Total number of SZ layers with sheet piling   |
| SpecifiedXYLevels = 1  | 0: not specified.<br>1: top and bottom levels specified for each layer<br>Note: only checked when FileVersion > 1         |
| [Layer_1]  | This section must be repeated for each -NrOfLayers- sheet piling layer. The sections must be named Layer_1, Layer_2, etc. |
| LayerNumber = 1  | The MIKE SHE SZ layer number of the actual sheet piling layer (1 = top layer).  |



| Line item   | Comment   |
|---|---|
| <pre> [GridCodes]   Type = 1   FixedValue = 0 [DFS_2D_DATA_FILE]   FILE_NAME =  .\SPGrid_1.dfs2    ITEM_COUNT = 1   ITEM_NUMBERS = 1 EndSect // DFS_2D_DATA_FILE EndSect // GridCodes </pre>  | <p><b>[GridCodes] section</b> Specification of grid codes for the current layer.</p> <p><b>Type</b> Normally 1 because a dfs2 file is required. 0 means global value.</p> <p><b>FILE_NAME</b> Name of the dfs2 file with grid codes. The file name is enclosed in " " which tells the system that the name is relative to the location of this module input file.</p> <p><b>ITEM_NUMBERS</b> : One number (because ITEM_COUNT must be 1) defining the item of the dfs2 file to be used.</p> |
| <pre> [X_Leakage]   Type = 0   FixedValue = 1.0E-7 [DFS_2D_DATA_FILE]   FILE_NAME =  .\maps\SPLeakX_1.dfs2    ITEM_COUNT = 1   ITEM_NUMBERS = 1 EndSect // DFS_2D_DATA_FILE EndSect // X_Leakage </pre>   | <p><b>[X_Leakage] section</b> Required if there are any cells with N-S sheet piling affecting the flow in the x-direction (codes containing 100).</p> <p><b>Type</b> Set to 0 if a global value is specified and 1 if using a dfs2 file.</p> <p><b>FixedValue</b> The global value (1/s) which is read if Type = 0.</p> <p><b>FILE_NAME</b> and <b>ITEM_NUMBERS</b> Dfs2 file name and item number if Type = 1 (relative file name as explained under Grid Codes).</p>                      |
| <pre> [Y_Leakage]   Type = 0 //(0:Fixed value,1:DFS2 file)   FixedValue = 2.0E-7 [DFS_2D_DATA_FILE]   FILE_NAME =  .\maps\SPLeakY_1.dfs2    ITEM_COUNT = 1 //(must be 1)   ITEM_NUMBERS = 1 1 EndSect // DFS_2D_DATA_FILE EndSect // Y_Leakage </pre> | <p><b>Y_Leakage] section</b> : Required if there are any cells with E-W sheet piling affecting the flow in the y-direction (codes containing 10).</p>   |



| Line item   | Comment  |
|---|--|
| [Z_Leakage]<br>Type = 0 //(0:Fixed value,1:DFS2 file)<br>FixedValue = 3.0E-7<br>[DFS_2D_DATA_FILE]<br>FILE_NAME =  .\maps\SPLeakZ_1.dfs2 <br>ITEM_COUNT = 1<br>ITEM_NUMBERS = 1<br>EndSect // DFS_2D_DATA_FILE<br>EndSect // Z_Leakage  | [Z_Leakage] section : Required if there are any cells with horizontal sheet piling affecting the vertical flow (codes containing 1). |
| [X_TopLevel]<br>RelativeToGround = 0 // 0: no, 1: yes<br>Type = 1 //(0:Fixed value,1:DFS2 file)<br>FixedValue = 0.0<br>[DFS_2D_DATA_FILE]<br>FILE_NAME =  .\YLevels_1.dfs2 <br>ITEM_COUNT = 1 //(must be 1)<br>ITEM_NUMBERS = 1<br>EndSect // DFS_2D_DATA_FILE<br>EndSect // Y_TopLevel       | [X_TopLevel] section : Required if SpecifiedXYLevels=1 and there are any codes containing 100.                                       |
| [X_BottomLevel]<br>RelativeToGround = 0 // 0: no, 1: yes<br>Type = 1 //(0:Fixed value,1:DFS2 file)<br>FixedValue = 0.0<br>[DFS_2D_DATA_FILE]<br>FILE_NAME =  .\YLevels_1.dfs2 <br>ITEM_COUNT = 1 //(must be 1)<br>ITEM_NUMBERS = 2<br>EndSect // DFS_2D_DATA_FILE<br>EndSect // Y_BottomLevel | [X_BottomLevel] section : Required if SpecifiedXYLevels=1 and there are any codes containing 100.                                    |



| Line item   | Comment  |
|---|--|
| [Y_TopLevel]<br>RelativeToGround = 0 // 0: no, 1: yes<br>Type = 1 //(0:Fixed value,1:DFS2 file)<br>FixedValue = 0.0<br>[DFS_2D_DATA_FILE]<br>FILE_NAME =  .\YLevels_1.dfs2 <br>ITEM_COUNT = 1 //(must be 1)<br>ITEM_NUMBERS = 1<br>EndSect // DFS_2D_DATA_FILE<br>EndSect // Y_TopLevel       | [Y_TopLevel] section : Required if SpecifiedXYLevels=1 and there are any codes containing 10.    |
| [Y_BottomLevel]<br>RelativeToGround = 0 // 0: no, 1: yes<br>Type = 1 //(0:Fixed value,1:DFS2 file)<br>FixedValue = 0.0<br>[DFS_2D_DATA_FILE]<br>FILE_NAME =  .\YLevels_1.dfs2 <br>ITEM_COUNT = 1 //(must be 1)<br>ITEM_NUMBERS = 2<br>EndSect // DFS_2D_DATA_FILE<br>EndSect // Y_BottomLevel | [Y_BottomLevel] section : Required if SpecifiedXYLevels=1 and there are any codes containing 10. |
| EndSect // Layer_1  |  |
| EndSect // SheetPiling  |  |
| EndSect // MIKESHE_SheetPiling_File   |  |

## 9.2 Negative Precipitation

Negative precipitation is sometimes required when net groundwater recharge has been calculated using an external program, such as DAISY GIS. In this case, the evapotranspiration may exceed infiltration leading to a net upward flux of water from the groundwater table. However, the standard precipitation module in MIKE SHE does not recognize negative rainfall. In this case, you must specify the negative rainfall using the following Extra Parameters options:

| Parameter Name   | Type    | Value             |
|--|---------|-------------------|
| use negative precipitation                                     | Boolean | On                |
| <i>If the negative precipitation is uniformly distributed:</i> |         |                   |
| negative precipitation max depth                               | float   | greater than zero |
| negative precipitation max layer                               | integer | greater than zero |



| Parameter Name   | Type      | Value  |
|--|-----------|--|
| <i>If the negative precipitation is spatially distributed:</i> |           |  |
| negative precipitation max depth<br>dfs2 file                  | file name | .dfs2 file                                     |
| negative precipitation max depth<br>dfs2 item                  | integer   | item number in dfs2 file,<br>greater than zero |
| negative precipitation max layer<br>dfs2 file                  | file name | dfs2 file                                      |
| negative precipitation max layer<br>dfs2 item                  | integer   | item number in dfs2 file,<br>greater than zero |

**Max depth** - This represents the depth of the root zone plus the thickness of the capillary fringe and is the maximum depth from which negative precipitation can be extracted.

**Max layer** - This is the maximum layer depth from which negative precipitation can be extracted.

### 9.3 *Precipitation Multiplier*

To facilitate calibration and sensitivity analysis of recharge, in models where measured precipitation is not being used, a multiplication factor has been implemented.

| Parameter Name       | Type  | Value             |
|----------------------|-------|-------------------|
| precipitation factor | float | greater than zero |

If this extra parameter is used, then all precipitation values are multiplied by the factor prior to being used in MIKE SHE.

### 9.4 *SZ Drainage to Specified MIKE 11 H-points*

This section details how to route drainage from the saturated zone drains and paved area runoff directly to MIKE 11 H-points within a specified range of chainage values for a specified reach. This option is called the Reference Drainage (RFD) option. The RFD option is different from the original drainage function that distributed drainage and paved area discharges along river links rather than directly at H-points. Furthermore, this



option can be used with MIKE 11 branches that are not defined in the MIKE SHE coupling section of the MIKE 11 network file.

The water balance utility (e.g., Saturated zone - detailed) can be used to look at differences between drainage discharges from areas using the original drainage option and the RFD option. The MIKE SHE water balance configuration file (MSHE\_Wbl\_Config.pfs in the Program Files\DHI\MIKEZero\bin subdirectory) should be reviewed to see which water balance types segregate standard drainage flow (data type sz.qszdr-torivin) and RFD drainage flow (data type sz.qszdrtoM11Hpoint).

The following steps are required to activate the RFD option:

- 1 Generate a pfs file containing information for each specified drainage area to be routed to specific MIKE 11 h-points using the command line program MakeSM11RFDpfs.exe. The program requires a tab-delimited file with the format shown below:

```
BRS_ID  BRS_START  BRS_END  Drain_code
```

The file expects a header line in the tab-delimited file and will ignore the first drainage area if a header is not included. The program will query you for the name of the tab-delimited input file and the name of the RFD pfs file to create. If program execution is successful the program will indicate that it has terminated normally. If the program indicates an abnormal termination has occurred you should review your tab-delimited file to make sure there are no errors in the input file. The most common error is use of a space-delimited file that contains branch names with spaces in the names.

**BRS\_ID** - The first column is the MIKE 11 branch name. The branch name must be spelled correctly and include all spaces contained in the name, if any. The branch name should not be enclosed in quotes. An error condition will occur if the specified branch is not present in the MIKE 11 network.

**BRS\_START** and **BRS\_END** - The second and third column is the starting and ending chainage of the specified branch (column one) which drainage and/or paved area discharge is routed to. The interval does not have to correspond exactly to specific MIKE 11 H-points because the MIKE SHE pre-processor finds the closest H-points to the specified interval. However, BRS\_START must be greater than or equal to BRS\_END. If BRS\_START and BRS\_END are the same the all drainage and/or paved area discharge is routed to the closest H-point.



**Drain\_code** - The fourth column specifies the drain code for the area that drainage and/or paved area discharge is routed to the specified MIKE 11 branch and chainage. The drain code must be greater than or equal to zero. Drain code values equal to zero (0) are not included in the reference drainage system. Furthermore, an error condition will occur if the specified drain code does not exist in the drainage code file used in MIKE SHE.

- 2 Navigate to the Extra Parameters entry on the menu tree and add the following items to the extra parameters list

| Parameter Name                     | Type      | Value   |
|------------------------------------|-----------|---|
| use specified reaches for drainage | Boolean   | On  |
| specified reaches for drainage     | file name | the correct file name, including the path, of the file created by the program MakeSM11RFDpfs.exe. |

- 3 Navigate to the Drainage item under the Saturated Zone on the menu tree and select **distributed drainage options**. See Drainage (V.2 p. 123).
- 4 Specify drain codes in the same manner as usual. Remember that all drain codes in the RFD option pfs file must exist in the active domain of the model or you will get an error.
- 5 With the RFD option, a drainage distribution must be defined. If the RFD is going to be used throughout the active model domain a uniform value of three can be specified for the drainage distribution. If a combination of the original drainage method and the RFD option is going to be used, a value of two and three should be used for areas using the original drainage option and the RFD option, respectively.
- 6 Pre-process and run your MIKE SHE model

If the MIKE SHE setup does not successfully preprocess you should review the above steps to see if you have any error in the setup. The *projectname*\_PreProcessor\_Messages.log file (where *projectname* is the name of your \*.she file) in your simulation subdirectory should help you identify why the MIKE SHE setup failed to preprocess.

If the MIKE SHE setup successfully preprocesses you should also look at the preprocessed data (on the Processed data tab) and the YourSetup\_PreProcessor\_Print.log file in your simulation subdirectory to make sure you are comfortable with how the preprocessor has set up the



drainage reference system. You can search for *Making setup of Specified MIKE 11 Reaches For Drainage* in the `YourSetup_PreProcessor_Print.log` file to find the start of the section that details the drainage reference system.

## 9.5 Time varying drainage parameters

In projects where you want to simulate the build out of a drainage network over time, or changes in the drainage time constants over time, then you can use this set of extra parameters. Without this set of extra parameters you would have to hot start your simulation at regular time intervals with the new drainage parameters.

The time varying drains are also allowed to shift between layers. However, if the drainage level goes above or below the model, the level will be adjusted and a warning is issued to the log file.

**Note:** The SOR solver does not allow drainage in any layer except the top layer and the drain level will be adjusted accordingly.

**Note:** If you specify time varying drainage parameters, you will not be able to use any of the drainage routing methods that depend on the drain level. The preprocessor checks this and gives an error if you have specified

- option 1 (routing based on levels), or
- option 3 (distributed options) AND any of the distributed option codes are 1 (routing based on levels in these cells).

To activate time varying drainage parameter options, you must specify the following extra parameters

| Parameter Name                             | Type      | Value                                       |
|--|-----------|---|
| time varying drainage levels               | Boolean   | On  |
| time varying drainage constants            | Boolean   | On  |
| time varying drainage level dfs2 file name | file name | .dfs2 file                                  |
| time varying drainage level item number    | integer   | item number in dfs2 file, greater than zero |





| Parameter Name   | Type      | Value                                       |
|--|-----------|---|
| time varying drainage time constant dfs2 file name                               | file name | .dfs2 file                                  |
| time varying drainage time constant item number                                  | integer   | item number in dfs2 file, greater than zero |
| <i>Optional if mean step accumulated values instead of instantaneous values:</i> |           |   |
| mean step accumulated drainage levels  | Boolean   | On  |
| mean step accumulated drainage time constants                                    | Boolean   | On  |

The dfs2 Drain Level is an elevation that can be specified using the following three EUM Data Units (*V.1 p. 349*):

- Elevation
- Depth Below Ground (i.e. positive values)
- Height Above Ground (i.e. negative values)

By default, the Time Series Types (*V.1 p. 244*) is Instantaneous, but there is an extra option that allows you to use Mean Step Accumulated values if you want.

**Note** The code does not check for the time series type.

All specifications are printed to the *projectname\_PreProcessor\_Print.log* and *projectname\_WM\_Print.log* files.

## 9.6 Irrigation River Source Factors

A global “river source volume factor” and “river source discharge factor” are available as extra parameters for increased control of river sources during irrigation.

| Parameter Name                | Type  | Value         |
|-------------------------------|-------|---------------|
| river source volume factor    | float | positive      |
| river source discharge factor | float | 0 or positive |



None, one, or both can be specified. If the factor is not specified, then a Volume factor of 0.99 and a Discharge factor of 0.0 will be used.

The factors are used in the calculation of the available water (depth) of a river source:

$$Depth = MIN\left(\frac{C_s \cdot \Delta t}{A}, \frac{F_V \cdot V_L}{A} + \frac{F_D \cdot D_L \cdot \Delta t}{A}\right) \quad (9.1)$$

where *Depth* is the available water depth in the river link,  $C_s$  is the source capacity,  $\Delta t$  is the time step length,  $F_v$  is the specified volume factor,  $V_L$  is the volume of water in the link,  $F_D$  is the specified volume discharge,  $D_L$  is the river link discharge, and  $A$  is the cell area.

The river link discharge is the same as used when checking with the threshold discharge for switching on/off the source. It is the absolute discharge in the middle of the MIKE SHE river link, interpolated between two MIKE 11 H-points.

MIKE SHE prints the following message in the xxx\_WM\_Print.log file when the parameters are specified:

```
Extra-parameter specified:  
river source volume factor  
value = 1.500000
```

```
Extra-parameter specified:  
river source discharge factor  
value = 1.000000
```

MIKE SHE also prints the following warnings in the xxx\_WM\_Init\_Messages.log file if one or both of the factors may result in water balance errors or numerical instabilities

```
WARNING: Specified value for river source volume factor  
is greater than 1 : 1.500000.  
There is a risk of water balance errors and/or insta-  
bilities in the coupling between MIKE SHE and MIKE 11.
```

```
WARNING: Specified value for river source discharge  
factor is greater than 0 : 1.000000.  
There is a risk of water balance errors and/or insta-  
bilities in the coupling between MIKE SHE and MIKE 11.
```



## 9.7 Distributed Snow Melt Constants

Distributed snow melt constants allows you spatially adjust the degree-day factor and melting temperature. To activate the distributed option you must add the following items to the list of extra parameters:

| Parameter Name                  | Type      | Value                                       |
|---------------------------------|-----------|---|
| distributed degree-day factor   | Boolean   | On  |
| degree-day factor dfs2 file     | file name | .dfs2 file; Type: Melting Coefficient       |
| degree-day factor dfs2 item     | integer   | item number in dfs2 file, greater than zero |
| distributed melting temperature | Boolean   | On  |
| melting temperature dfs2 file   | file name | dfs2 file; Type: Temperature                |
| melting temperature dfs2 item   | integer   | item number in dfs2 file, greater than zero |

The dfs2 file containing the degree-day factors must have the Type “Melting Coefficient”. The dfs2 file containing melting temperatures must have the Type “Temperature”.

Both the degree-day factor and the melting temperature are used in the so-called UZ Classification when you use multi-layer UZ (Gravity or Full Richards) and specify Automatic or Partial Automatic classification type.

## 9.8 Canyon exchange option for deep narrow channels

In the case of a deep, narrow channel crossing multiple model layers, the head difference used in Equations (14.1) and (14.2) can optionally be limited by the bottom elevation of the layer. Thus,

$$\Delta h = h_{grid} - \max(h_{riv}, z) \quad (9.2)$$

where  $z$  is the bottom of the current layer.



The above formulation reduces the infiltration from upper layers by reducing the available gradient. Without the 'Canyon' option, MIKE SHE effectively assumes that the river is hydraulically connected to the upper most model layer, since MIKE SHE calculates the exchange flow with all layers that intersect the river based on the difference between the river level and the water table.

Currently, this option is only available for steady-state models.

| Parameter Name         | Type    | Value |
|------------------------|---------|-------|
| enable canyon exchange | Boolean | On    |

## 9.9 Simplified Overland Flow Options

### Avoiding the redistribution of ponded water

In the standard version of the Simplified Overland Flow solver, the solver calculates a mean water depth for the entire flow zone using the available overland water from all of the cells in the flow zone. During the Overland flow time step, ET and infiltration are calculated for each cell and lateral flows to and from the zone are calculated. At the end of the time step, a new average water depth is calculated, which is assigned to all cells in the flow zone.

In practice, this results in a redistribution of water from cells with ponded water (e.g. due to high rainfall or low infiltration) to the rest of the flow zone where cells potentially have a higher infiltration capacity. To avoid this redistribution, an option has been added where the solver only calculates overland flow for the cells that can potentially produce runoff, that is, only in the cells for which the water depth exceeds the detention storage depth.

| Parameter Name             | Type    | Value |
|----------------------------|---------|-------|
| only simple OL from ponded | Boolean | On    |

### Routing simple overland flow directly to the river

In the standard version of the Simplified Overland Flow solver, the water is routed from 'higher' zones to 'lower' zones within a subcatchment. Thus, overland flow generated in the upper zone is routed to the next lowest flow zone based on the integer code values of the two zones. In other



words, at the beginning of the time step the overland flow leaving the upper zone (calculated in the previous time step) is distributed evenly across all of the cells in the receiving zone. In practice, this results in a distribution of water from cells in the upstream zone with ponded water (e.g. due to high rainfall or low infiltration) to all of the cells in the downstream zone with potentially a large number of those cells having a higher infiltration capacity. In this case, then, overland flow generated in the upper flow zone may never reach the stream network because it is distributed thinly across the entire downstream zone.

To avoid excess infiltration or evaporation in the downstream zone, an option was added that allows you to route overland flow directly to the stream network. In this case, overland flow generated in any of the overland flow zones is not distributed across the downstream zone, but rather it is added directly to the MIKE 11 stream network as lateral inflow.

| Parameter Name       | Type    | Value |
|----------------------|---------|-------|
| no simple OL routing | Boolean | On    |

## 9.10 2-Layer UZ Options

### 9.10.1 Threshold depth for infiltration

The 2-Layer water balance method for the unsaturated zone does not include evapotranspiration from the soil surface. Thus, even a small amount of water on the ground surface will infiltrate. If you use this extra parameter, then you can define a depth of overland water that must be exceeded before infiltration will occur. This keeps small amounts of precipitation from infiltrating and allows them to evaporate instead.

The calculated infiltration is simply reduced if the remaining overland water depth will be smaller than the specified threshold value.

| Parameter Name                         | Type    | Value             |
|--|---------|-------------------|
| use threshold depth for infiltration   | Boolean | On                |
| threshold depth for infiltration meter | Float   | Greater than zero |



### 9.10.2 Increase infiltration to dry soils

In dry soils the rate of infiltration can be higher than the saturated hydraulic conductivity because capillarity will draw water into the soil and increase the rate of infiltration. The Increase Infiltration to Dry Soils extra parameter is available to account for this process, when Richards equation is not being used.

If the actual water content in the root zone is below the field capacity,  $\theta_{fc}$ , then the infiltration capacity is calculated as

$$K_{infiltration} = K_{infiltration} \cdot InfiltrationFactor$$

if

$$\theta_{actual} > \frac{\theta_{fc} - \theta_{wp}}{InfiltrationFactor} + \theta_{wp}$$

where  $\theta_{wp}$  is the wilting point water content.

Otherwise, the infiltration capacity is calculated as

$$K_{infiltration} = K_{infiltration} \cdot \left( \frac{\theta_{fc} - \theta_{wp}}{\theta_{actual} - \theta_{wp}} \right)$$

..

| Parameter Name                     | Type    | Value            |
|------------------------------------|---------|------------------|
| increase infiltration to dry soils | Boolean | On               |
| max infiltration rate factor       | Float   | Greater than 1.0 |

## 9.11 GeoViewer Output

The GeoViewer is a MIKE Zero tool that is used in the MIKE GeoModel product for viewing geologic cross-sections in your conceptual model.

The GeoViewer Output extra parameters will create a set of dfs2 output files during the pre-processing that will allow you to look at your pre-processed model in the GeoViewer.



The GeoViewer Output is activated by

| Parameter Name           | Type    | Value |
|--------------------------|---------|-------|
| make SZ level dfs2 files | Boolean | On    |
| <i>Optional</i>          |         |       |
| adjust dfs2 levels       | Boolean | On    |

If this option is active, then the following files will be created:

- *setupname\setupname\_GeoLayers.dfs2* - containing the top and bottom of each geologic layer

If there are lenses:

- *setupname\setupname\_GeoLenses.dfs2* - containing the top and bottom of each geologic lense and delete values where there are no lenses

If the computational layers are not defined by geologic layers:

- *setupname\setupname\_CompLayers.dfs2* - containing the top and bottom of each computational layer

If the optional second parameter is used, then the top and bottom elevations that are written to the files will be adjusted to be confined between the topography and the lowest computational layer.







*dh Software*

**SURFACE WATER MODELLING  
WITH MIKE 11**





## 10 COUPLING MIKE 11 AND MIKE SHE

### 10.1 MIKE SHE

In the Rivers and Lakes dialogue (see Figure 10.1) you can link MIKE SHE to a MIKE 11 model. Choosing Edit in this dialogue, will open the MIKE 11 simulation (.sim11) file.

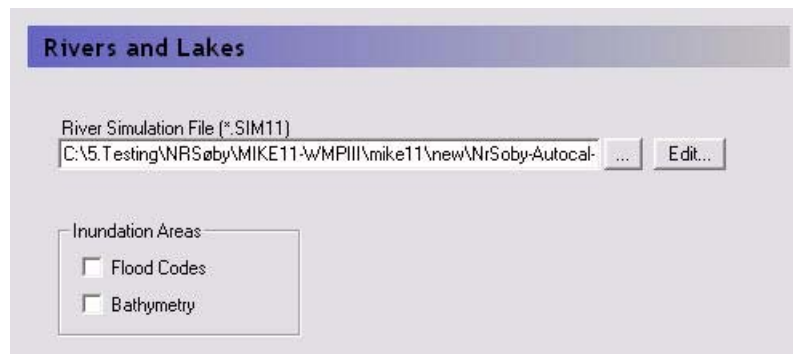


Figure 10.1 Rivers and Lakes dialogue for specifying the MIKE 11 .sim11 file

The River Simulation File (.sim11) is the main MIKE 11 simulation file, which contains the file references to all the files used in the MIKE 11 model. for MIKE SHE, this is primarily the river network file (.nwk11), the cross-section database (.xns11), the boundary condition file (.bnd11) and the hydrodynamic setup file (.hd11).

In the Rivers and Lakes dialogue, there are two Inundation Areas options. These options are always available for input, but are only used if you have selected specific options in the MIKE SHE Links dialogue (Figure 10.2) for calculating inundation areas. These options are

- Flood codes - used for the Area Inundation using Flood Codes (areal source/sink) (V.2 p. 239) option in MIKE 11, and
- Bathymetry - used to modify the defined topography with a more detailed flood plain topography in areas where Flood Codes have been defined.

Integrating a MIKE SHE and a MIKE 11 model is not very different from establishing a stand-alone MIKE 11 HD model and a stand-alone MIKE SHE model. In principle there are three basic set-up steps:



- 1 Establish a MIKE 11 HD hydraulic model as a stand-alone model and make a performance test and, if possible, a rough calibration using prescribed inflow and stage boundaries. You can also specify a default groundwater table (e.g. MIKE SHE's initial groundwater level) and leakage coefficients for any leakage calculations.
- 2 Establish a MIKE SHE model that includes the overland flow component and (optionally) the saturated zone and unsaturated zone components. An SZ drainage boundary can be used to prevent excessive surface flows in low lying areas and the river flood plain.
- 3 Couple MIKE SHE and MIKE 11 by defining branches (reaches) where MIKE 11 HD should interact with MIKE SHE. Modify your MIKE SHE and MIKE 11 models so that they work together properly. For example, by removing the specified groundwater table in MIKE 11 and adjusting your SZ drainage elevations if you used these in Step 2.

With regard to Step 1, a normal MIKE 11 model is set-up, which is described in the MIKE 11 documentation and on-line help facilities. However, a few extra hints are given in Section 10.2. Step 2 is described elsewhere in this manual. The rest of this chapter is largely about Step 3.

## **10.2 MIKE 11**

Typically, the first step in coupling MIKE 11 to MIKE SHE is to create a normal MIKE 11 HD model without coupling it with MIKE SHE. In this regard, a few things should be emphasised:

- In a normal MIKE 11 river model only the river chainage (dx) is important for the results. Geographic positioning of river branches and cross-sections are only important for the graphical presentation. When interfacing MIKE 11 to MIKE SHE geographic positioning is critical, as MIKE SHE needs information on the river location.
- A reasonably high number of river cross-sections should be included to ensure that the river elevations are reasonably consistent with the surface topographic features.

### **10.2.1 MIKE 11 Cross-sections**

Whenever there is a significant change in the bed slope there should, in principle, be a cross-section defined in MIKE 11. If only a few cross-sections are available, it may be sufficient to estimate the cross-section shape based on neighbouring cross-sections and estimate the bank/bed elevation based on the surface topographic information in MIKE SHE or other topographic maps.



However, every cross-section in MIKE 11 is a calculation node. The time step in MIKE 11 is sensitive to the Courant number, which is proportional to the distance between calculation nodes. So, if the cross-sections are close together, then you may experience very short time steps in MIKE 11.

Thus, if you have very short MIKE 11 time steps, then you might want to check your river network to make sure you do not have cross-sections that are too close together. This frequently occurs when the cross-sections have been imported. If you do have cross-sections that are too close together, then you can easily eliminate one or more of them, as long as the conveyance of the different cross-sections is roughly the same. In other words, you can eliminate duplicate cross-sections if their  $Q/H$  relationships are roughly the same, even though the physical shape of the two cross-sections may appear quite different. This is often the case in braided stream networks, where the location of the main channels may move left or right, but the overall conveyance of the river bed is relatively constant.

### Cross-sections versus MIKE SHE topography

In the absence of flooding, ponded water discharges to the MIKE 11 river as overland flow. As a general rule, the topography must be higher than or equal to the bank elevation. If the bank elevation is higher than the topography, water will not be able to flow into the river in that cell, but will run laterally along the river until it reaches a place for it to flow into the river. An easy trick to see where this is happening is to run a simulation with no infiltration, ET, or detention storage and set the initial water depth at 1m. Then look at the results to find places where the water is piling up against the river links.

In the pre-processor log file, a table is created that lists all the river links where the bank elevation is different than the topography of the adjacent cell. The critical river links with bank elevations above the topography are highlighted with the ==> symbol. This list can be surprisingly long because the river link bank elevations are interpolated from the neighbouring cross-sections. Whereas the topography is already defined. So, frequently the interpolated bank elevations do not line up precisely with the topography.

If overland flow on the flood plain is essentially absent, for example, due to infiltration or evapotranspiration, then these differences are not relevant and there is no need to modify the topography. However, if the overland to river exchange is important then you may have to carefully modify your topography file or your bank elevations so that they are consistent.

**Hint** In many cases, your topography is from a DEM that is different from your model grid - either because it is a .shp or xyz file, or if it is a different



resolution than your model grid. In this case, it may be easier to save the pre-processed topography to a dfs2 file (right click on the topography map in the pre-processed tab). Then modify and use the new dfs2 file as the topography in your model setup. The disadvantage of this, is that if you change your model domain or grid, then you will have to redo your topography modifications.

**Hint** You can also use one of the Flood code options to automatically modify your topography, if you have wide cross-sections or a detailed bathymetry file. In this case, after you have set up your MIKE 11 model you can specify a constant grid code for the whole model and let MIKE SHE calculate a modified topography based on the cross-sections or bathymetry. Then save the topography file as above and then use it as the model topography.

### **Flooding with Overbank Spilling**

If you are simulating flooding on the flood plain using the Direct Overbank Spilling to and from MIKE 11 (*V.2 p. 240*) option, then the MIKE 11 cross-sections are normally restricted to the main channel. The flood plain is defined as part of the MIKE SHE topography. Since, the bank elevation is used to define when a cell floods, it is more critical that the cross-sections are consistent with your topography, in the areas where you want to simulate flooding. The table in the simulation log file mentioned above is useful to locate these inconsistencies. It is usually necessary to have a very fine grid and a detailed DEM for such simulations, which tends to reduce the inconsistencies because it reduces the amount of interpolation and averaging when creating the model topography.

### **Flooding with Flood Codes**

If you are simulating flooding on the flood plain using the Area Inundation using Flood Codes (areal source/sink) (*V.2 p. 239*) option, then flood plain elevation should be consistent with the cross-sections. Otherwise, the flood plain storage will be inconsistent with the river storage based on the cross-sections.

When you are using Flood Codes, you typically specify wide cross-sections for your rivers. The wide cross-sections can then account for the increased flood plain storage during flood events. MIKE 11 then places water on the MIKE SHE cells that are defined by flood codes - if the water level in the river is above the cell topography. The flood water is then free to infiltrate or evaporate as determined by MIKE SHE.

In such flooded cells, overland flow is no longer calculated, so there is no longer any overland exchange to MIKE 11 in flooded cells. Thus, the bank elevation is not so critical, as long as the cell is flooded. However, when



the flood recedes, the cells revert back to normal overland flow cells and the same considerations apply as if the cells were not flooded - namely the bank elevation should be below the topography to ensure that overland flow can discharge to the river link.

Flood codes are also commonly used for lakes and reservoirs. In this case, you specify the lake bed bathymetry as the topography (or using the Bathymetry option). The lake area is defined using flood codes and the MIKE 11 cross-sections stretch across the lake. MIKE 11 calculates the lake level and floods the lake. Overland flow adjacent to the lake intersects the flooded cells and the overland water is added to the lake cell (and to MIKE 11 as lateral inflow). Groundwater exchange to the lake is through the lake bed as saturated zone discharge. In principle, the saturated zone could discharge to the river link, but the local groundwater gradients would probably make this exchange very small.

### **Combining Flood Codes and Overbank Spilling**

Flooding using Overbank spilling and Flood Codes is possible in the same model and even in the same coupling reach. The only restriction is that there is no overland flow calculated in cells flooded by means of Flood Codes. So, in a long coupling reach, you could allow overbank spilling and calculate overland flow using the explicit solver, but define flood codes in the wide downstream flood plain where the surface water gradients are very low during flooding and in the wide shallow reservoir half way down the system.

## **10.2.2 MIKE SHE Coupling Reaches**

Each MIKE 11 branch that exchanges water with MIKE SHE is called a coupling reach. A MIKE 11 branch can be sub-divided into several coupling reaches. A reason for doing so could be to allow different riverbed leakage coefficients for different parts of the river.

Each coupling reach is interpolated to a MIKE SHE river link, which is located at the nearest boundary between grid cells. Each MIKE SHE river link can only be associated with one coupling reach, which restricts the coupling reaches from being too close together. This can lead to problems when you have a detailed drainage or river network with branches less than one half a cell width apart. If you have coupling reaches that are too close together, you will receive an error message. If this happens, you can

- decide not to include one of the branches as a coupling reach (it is still included in the MIKE 11 HD model), or
- remove some of the branches (this error often occurs when you have a detailed looped drainage network), or



- refine your MIKE SHE grid until all coupling reaches are assigned to unique river links.

The coupling reach-river link system is a conceptual representation of the river location in the model space that is grid dependent. The river course will be more accurately represented in a finer model grid.

If you have a regional model with large cells (say 1-2km wide), then you cannot expect the river-aquifer interaction to be accurate at the individual cell level (e.g. all your cell properties – topography, conductivity, ManningsM, etc. – are all average values over 1-4 km<sup>2</sup>). Rather, most often you will be interested in having a correct overall water balance along the stream. Typically, this is achieved by calibrating a uniform average river bed leakage coefficient against a measured outflow hydrograph. In such a model, you may also be tolerant of groundwater residuals on the order of a meter or more.

On the other hand, if you need more detailed site specific results (and you have data and measurements to calibrate against), then you will use a local scale model, with a smaller grid (say 50-200m) and discrepancies between topography and river bank elevation will largely disappear. In this case, you will be more likely to be able to make accurate local scale predictions of groundwater-surface water exchange.

The upper half of the dialogue displays the properties of the current coupling reach. While, the bottom half of the dialogue is a table listing all of the coupling reaches defined.





**Location**

Branch name:

Upstream Chainage:

Downstream Chainage:

**River-aquifer exchange**

Conductance:

Leakage Coef.:

**Weir data for overland-river exchange**  
(Select weir option in MIKE SHE)

Weir coefficient:

Head exponent:

Minimum upstream height above bank for full weir width:

☒ Allow overbank spilling

Minimum flow area for overbank spilling:

**Inundation options by Flood Code**

Flood Area Option:

Flood Code:

Bed Topography:

Bed Leakage:

**Overview of MIKE SHE Coupling Reaches**

|    | Branch Nam  | US. Chainag | DS. Chainag | Conductance    | Leakage Co | Flood Area  | Flood Code | Be  |
|----|-------------|-------------|-------------|----------------|------------|-------------|------------|-----|
| 1  | AmericanC   | 0           | 7900        | Aquifer + Bed  | 1E-006     | No flooding |            | Use |
| 2  | ArroyoCre   | 0           | 4019        | River bed only | 1E-006     | No flooding |            | Use |
| 3  | BaleSlough  | 0           | 6039        | River bed only | 1E-006     | No flooding |            | Use |
| 4  | BearCreek   | 0           | 6041        | River bed only | 1E-006     | No flooding |            | Use |
| 5  | BellaOaks   | 0           | 3164        | River bed only | 1E-006     | No flooding |            | Use |
| 6  | BellCreek   | 0           | 7090        | River bed only | 1E-006     | No flooding |            | Use |
| 7  | BellCreekL  | 0           | 3196        | River bed only | 1E-006     | No flooding |            | Use |
| 8  | BellCreek-t | 0           | 2035        | River bed only | 1E-006     | No flooding |            | Use |
| 9  | BellReserv  | 0           | 935         | River bed only | 1E-006     | Manual      | 3          | Use |
| 10 | BennettCr   | 0           | 2618        | River bed only | 1E-006     | No flooding |            | Use |
| 11 | BiterCreek  | 0           | 2803        | River bed only | 1E-006     | No flooding |            | Use |
| 12 | BlossomCr   | 0           | 4217        | River bed only | 1E-006     | No flooding |            | Use |
| 13 | BrownsVall  | 0           | 6171        | River bed only | 1E-006     | No flooding |            | Use |
| 14 | BuhmanCr    | 0           | 8209        | River bed only | 1E-006     | No flooding |            | Use |

Figure 10.2 MIKE SHE River Links dialogue in the tabular view of the MIKE 11 Network Editor

### Include all branches button

If the Include all branches button is pressed all the branches in the MIKE 11 setup will be copied to the MIKE SHE Links table. Branches that should not be in the coupling can subsequently be deleted manually and the specifications for the remaining branches completed. Thus, you may have a large and complex hydraulic model, but only couple certain reaches to MIKE SHE. All branches will still be in the hydraulic MIKE 11 model but MIKE SHE will only exchange water with branch reaches that are listed in the MIKE SHE links table.

**Note** The Include all branches button will erase all existing links that have been specified.

### Location

The **branch name**, **upstream chainage** and **downstream chainage** define the stretch of river that can exchange water with MIKE SHE. A MIKE 11 branch can be sub-divided into several coupling reaches, to



allow, for example, different riverbed leakage coefficients for different parts of the river.

## River Aquifer Exchange

### Conductance

The river bed conductance can be calculated in three ways.

**Aquifer only** - When the river is in full contact with the aquifer material, it is assumed that there is no low permeable lining of the river bed. The only head loss between the river and the grid node is that created by the flow from the grid node to the river itself. This is typical of gaining streams, or streams that are fast moving. More detailed information on this option can be found in Aquifer Only Conductance (*V.2 p. 234*).

**River bed only** - If there is a low conductivity river bed lining, then there will be a head loss across the lining. In this case, the conductance is a function of both the aquifer conductivity and the conductivity of the river bed. However, when the head loss across the river bed is much greater than the head loss in the aquifer material, then the head loss in the aquifer can be ignored (e.g. if the bed material is thick and very fine and the aquifer material is coarse). This is the assumption used in many groundwater models, such as MODFLOW. More detailed information on this option can be found in River bed only conductance (*V.2 p. 235*).

**Aquifer + Bed** - If there is a low conductivity river bed lining, then there will be a head loss across the lining. In this case, the conductance is a function of both the aquifer conductivity and the total conductivity of the between the river and the adjacent groundwater can be calculated as a serial connection of the individual conductances. This is commonly the case, when the aquifer material presents a significant head loss. For example, when the aquifer is relatively fine and the groundwater cells are quite large. More detailed information on this option can be found in Both aquifer and river bed conductance (*V.2 p. 237*).

### Leakage Coefficient - [1/sec]

This is the leakage coefficient for the riverbed lining in units of [1/seconds]. The leakage coefficient is active only if the conductance calculation method includes the river bed leakage coefficient.

## Weir Data for overland-river exchange

The choice of using the weir formula for overland-river exchange is a global choice made in the MIKE SHE OL Computational Control Parameters (*V.2 p. 34*) dialogue. If the weir option is chosen in MIKE SHE, then all



MIKE 11 coupling reaches will use the weir formula for moving water across the river bank. The weir option is typically used when you want to simulate overbank spilling and detailed 2D surface flow in the flood plains. The following parameters and options are available when you specify the weir option in MIKE SHE. If you chose the Manning equation option in MIKE SHE, then these parameters are ignored.

### **Weir coefficient and Head exponent**

The Weir coefficient and head exponent refer to the C and k terms respectively in Equation (14.8). The default values are generally reasonable. Both the weir coefficient and the head exponent are dimensionless.

### **Minimum upstream height above bank for full weir width**

In Equation (14.8), when the upstream water depth above the weir approaches zero, the flow over the weir becomes undefined. To prevent numerical problems, the flow is reduced linearly to zero when the water depth is below the minimum upstream height threshold. The EUM data type is Water Depth.

### **Allow overbank spilling**

This checkbox lets you define which branches are allowed to flood over their banks. Thus, you can allow flooding from MIKE 11 only in branches with defined flood plains, or only in areas of particular interest.

If overbank spilling is not allowed for a particular branch, then the overland-river exchange is still calculated using the weir formula, but the exchange is only one way - that is from overland flow to the river.

### **Minimum flow area for overbank spilling**

The minimum flow area threshold prevents overbank spilling when the river is nearly dry. The flow area is calculated by dividing the volume of a water in the coupling reach by the length of the reach. The EUM data type is Flow Area.

## **Inundation options by Flood Code**

This section specifies the options available for the Area Inundation using Flood Codes (areal source/sink) (V.2 p. 239) method of flooding MIKE SHE grid cells from MIKE 11.

The Inundation method allows specified model grid cells to be flooded if the MIKE 11 water level goes above the topography of the cell. In this case, water from MIKE 11 is “deposited” onto the flooded cell. The flood water can then infiltrate, or evaporate. However, overland flow between flooded cells and to the river is not calculated. Also, the flooded water



remains as part of the MIKE 11 water balance and is only transferred to MIKE SHE when it infiltrates.

Inundation areas and their associated Flood codes are specified on a coupling reach basis.

### Flood Area Option

The following three options are available for the Flood Area Option:

- **No Flooding** (default) With the No flooding option, the MIKE 11 river is confined between the left and right banks. If the water level goes above the bank elevation, then the river is assumed to have vertical banks above the defined left and right bank locations. No flooding via flood codes will be calculated.

**Note** If neither inundation nor overbank spilling is allowed, then there is the overland flow exchange to the river is one way only. The only mechanism for river water to flow back into MIKE SHE is through baseflow infiltration to the groundwater. If overland flow does spill into the river, there is first a check to make sure that the water level in the river is not higher than the ponded water.

- **Manual** If the Manual option is selected, then you must supply a Flood code map in MIKE SHE. This Flood code map is used to establish the relationship between MIKE 11 h-points and individual model grids in MIKE SHE. MIKE SHE then calculates a simple flood-mapping during the pre-processing that is used during the simulation to assign river water stages to the MIKE SHE cells if the river level is above the topography.
- **Automatic** The automatic flood mapping option is useful if the river network geometry is not very complex or for setting up the initial flood mapping, for later refinement. The automatic method, maps out a polygon for each coupling reach based on the left and right bank locations of all the cross-sections along the coupling reach. All cells within this polygon are assigned an integer flood code, unique to the coupling reach. The automatic method works reasonably well along individual branches with cross-sections that represent the flood plain. At branch intersections the assigned flood code may not be correct. However, this is often not serious because at river confluences the water levels in the different branches are roughly the same anyway. In any case, the flood code map is available in MIKE SHE's preprocessed tab, where you can check its reasonableness. Right clicking on the map will give you the option of saving the map to a dfs2 file, which you can then correct and use with the Manual option.



### Flood Code

If the Manual option is selected, then you must specify a Flood code for the coupling reach. The flood code is used for mapping MIKE SHE grids to MIKE 11 h-points. You must click on the Flood Code checkbox in Figure 10.2, and then specify an integer flood code file in MIKE SHE. The specified flood code for the coupling reach must exist in the dfs2 Flood Code file. It is important to use unique flood codes to ensure correct flood-mapping.

### Bed Topography

Since the flood mapping procedure will only flood a cell when the river water level is above the cell's topography, accurate flood inundation mapping requires accurate elevation data. If one of the flood options are selected, then you have the option to refine the topography of the flood plain cells based on the actual cross-section elevations or on a more detailed local-scale DEM, if it exists.

- **Use Grid Data** (default) If Grid Data option is selected, the MIKE SHE topography value is used to determine whether or not the cell is flooded. However, the program first checks to see if a Bathymetry file has been specified (see Figure 10.1).

If a Bathymetry file is available, the topography values of the cells with flood codes are re-interpolated based on the bathymetry data. The bathymetry option is useful when a more detailed DEM exists for the flood plain area compared to the regional terrain model.

- **Use Cross-section** If the Cross-section option is specified the topography values of the cells with flood codes are re-interpolated based on the cross-section data.

When the cross-section option is selected, the pre-processor maps out a flood-plain polygon for the coupling reach, based on the left and right bank locations of all the cross-sections along the coupling reach. Interpolated cross-sections are created between the available actual cross-sections, if the cross-section spacing is greater than  $\frac{1}{2} \Delta x$  (grid size). All the cross-sections (real and interpolated) are sampled to obtain a set of point values for elevation in the flood plain. The topography values of all cells with the current flood code that are within the flood-plain polygon are re-interpolated using the bilinear interpolation method to obtain a new topography value.

In principle, the Cross-section option ensures a good consistency between MIKE SHE grid elevations and MIKE 11 cross-sections. There will, however, often be interpolation problems related to river meandering, tributary connections, etc., where wide cross-sections of



separate coupling reaches overlap. Thus, you can make the initial MIKE SHE set-up using the Cross-section option and then subsequently retrieve and check the resulting ground surface topography, from the pre-processed data. If needed, the pre-processed topography can be saved to a .dfs2 file (right click on the map), modified and then used as input for a new set-up, now using the Use Grid Data option.

### **Bed Leakage**

If one of the flood options are selected, then you must also specify if and how the leakage coefficient will be applied on the flooded cells. The infiltration/seepage of MIKE SHE flood grids is calculated as ordinary overland exchange with the saturated or unsaturated zone. That is, the leakage coefficient, if it exists, is applied to both saturated exchange to and from the flooded cell and unsaturated leakage from the flooded cell. In the case of the unsaturated leakage, the actual leakage is controlled by either the leakage coefficient or the unsaturated zone hydraulic conductivity relationship - which yields the lowest infiltration rate.

- **Use grid data** In this case, the leakage coefficient specified in Overland-groundwater Leakage Coefficient is used. If this item has not been specified, then the leakage coefficient will be calculated based on the aquifer material only.
- **Use river data** (default) In this case, the Leakage Coefficient - [1/sec] for the coupling reach is actually copied to the flooded cell and used for all flood grid points of the coupling reach. This makes sense if the flood plain is frequently flooded and covered with the same sediments as the river bed. However, in many cases the flood plain material is not the same as the river bed and the infiltration rate can be substantially different.

## **10.3 Converting from Flood Codes to Overbank Spilling**

The explicit solver and overbank spilling from MIKE 11 to overland flow are new in the 2007 Release. In principle, if you were careful setting up your flood codes, then the conversion to overbank spilling should result in the same flooded area, with similar depths. The only difference will be that the water on the flooded area is flowing.

However, in practice the conversion is not likely to be this smooth. Flood code setups are typically done manually and the topography is typically not very closely controlled - as long as it was inundated when it was supposed to be. Furthermore, the need for detailed surface roughness (Manning's  $n$ ) will require additional data. Finally, the complication of fully



dynamic (diffusive wave) 2D flow can lead to complicated water flows across the flood plain. So, there is likely to be substantial adjustment and re-calibration to get the flooding right.

Fortunately, you can mix Flood codes and Overbank spilling in the same model and even in the same coupling reach. This allows you to update only the parts of your model where the overbank spilling is important and leave the Flood code option intact elsewhere.

## **10.4 Common MIKE 11 Error Messages**

There are a number of common MIKE 11 error messages that you are likely to encounter when using MIKE 11 with MIKE SHE.

### **10.4.1 Error No 25: At the h-point \_\_\_\_\_ the water depth greater than 4 times max. depth**

This error message essentially says that your MIKE 11 model is unstable. It frequently occurs when there is an inconsistency in your bed elevations at the branch junctions. For example, if the bed elevation of the main branch is much greater than the side branch, then the water piles up and causes this error.

### **10.4.2 Warning No 47: At the h-point \_\_\_\_\_ the water level as fallen below the bottom of the slot x times**

This warning message essentially says that your MIKE 11 model is unstable. The slot is a numerical trick that keeps a very small amount of water in the MIKE 11 cross-section when the river is dry. So, when the water level falls below the slot, it implies that your river has dried out. This warning frequently occurs when there is either an inconsistency in your bed elevations or there is an error in your boundary conditions that is keeping water from entering the system.







*Phi Software*

**DRAINAGE MODELLING WITH  
MOUSE**





## 11 USING MIKE SHE WITH MOUSE

Coupling MOUSE and MIKE SHE allows you to simulate the effect of urban drainage and sewer systems on the surface/subsurface hydrology.

The use of the integrated MIKE SHE/MOUSE system is not very different from establishing a stand-alone MOUSE model and a stand-alone MIKE SHE model. In principle there are three basic set-up steps to have a coupled MIKE SHE-MOUSE model:

- 1 Establish a MOUSE hydraulic model as a stand-alone model, make a performance test and, if possible, a rough calibration using prescribed inflow and boundaries.
- 2 Establish a MIKE SHE model that includes the overland flow component and (optionally) the saturated zone and unsaturated zone components.
- 3 Couple MIKE SHE and MOUSE by defining the locations where MOUSE should interact with MIKE SHE.

When MIKE SHE runs, it will call MOUSE and ask it to perform a MOUSE time step. If the end of the MIKE SHE time step has not yet been reached MIKE SHE will ask MOUSE to calculate the next MOUSE time step. The MOUSE model will run normally if it is launched directly from MOUSE.

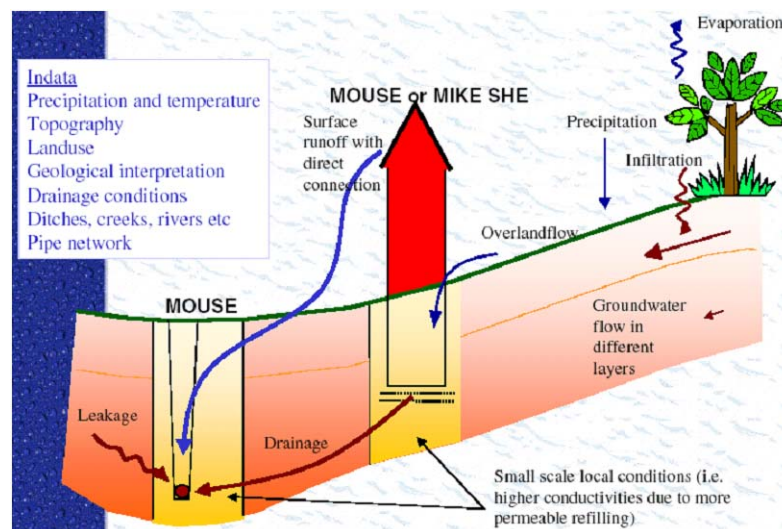


Figure 11.1 MIKE SHE to MOUSE coupling linkages



The exchange between MOUSE and MIKE SHE is calculate based on the following equation

$$Q = C \cdot (H_{SHE} - H_{MOUSE})^k \quad (11.1)$$

where  $Q$  is the exchange between MOUSE and MIKE SHE,  $C$  is the exchange coefficient,  $k$  is a head difference exponent and

$$H_{SHE} = \text{Max}(H_{cell}, Z_T, Z_M) \quad (11.2)$$

$$H_{MOUSE} = \text{Max}(H_{pipe}, Z_T, Z_M) \quad (11.3)$$

where  $H_{cell}$  is the head in the MIKE SHE cell,  $H_{pipe}$  is the head in the MOUSE pipe,  $Z_T$  is the topographic elevation in the cell and  $Z_M$  is the elevation of the manhole.

There are five variations on how to calculate the exchange based on above equations:

### MIKE SHE SZ to MOUSE LINKS

This is a leakage-based solution in which the head difference exponent is 1 and the exchange coefficient in Equation (11.1) for the flow to or from the pipe is calculated by

$$C = C_L \cdot R_H \cdot L \quad (11.4)$$

where  $C_L$  is the leakage coefficient (see below),  $R_H$  is the hydraulic radius for the flow (see below), and  $L$  is the length of the MOUSE pipe (link) in the MIKE SHE cell.

**Leakage Coefficient** - The leakage coefficient can be defined in two ways.

**Option 1** is the simple method, which is to use the pipe leakage coefficient specified in the MOUSE .ADP file. See Telling MOUSE that it is coupled to a MIKE SHE model (*V.1 p. 183*).

**Option 2** uses a combination of the pipe leakage coefficient and the aquifer hydraulic conductivity. In this case, the leakage coefficient is calculated as a series connection of the pipe leakage coefficient ( $C_p$ ) and the



“average” leakage coefficient of the aquifer grid cell ( $C_{aq}$ ). The average leakage coefficient of the grid cell is calculated assuming that the exchange of water between the pipe and the grid cell is both vertical and horizontal. The leakage coefficient calculation does not calculate a detailed flow path based on a geometric calculation, since a MOUSE pipe can be located anywhere in a grid cell. Instead, an average vertical and horizontal flow distance is used based on 1/4 of the vertical and horizontal cell dimensions. Thus,

$$C_{aq} = C_{aqH} + C_{aqV} = \frac{K_x}{(\Delta x)/4} + \frac{K_z}{(\Delta z)/4} \quad (11.5)$$

where  $K_x$  and  $K_z$  are the horizontal and vertical hydraulic conductivities respectively and  $\Delta x$  and  $\Delta z$  are the horizontal and vertical cell dimensions.

The final leakage coefficient is then calculated as the harmonic mean of both the aquifer leakage coefficient and the piper leakage coefficient:

$$\frac{1}{C_L} = \frac{1}{C_{aq}} + \frac{1}{C_p} \quad (11.6)$$

**Hydraulic Radius** - MIKE SHE uses the inner hydraulic radius if the flow is from MOUSE to MIKE SHE. Whereas, it uses the outer hydraulic radius if the flow is from MIKE SHE to MOUSE. The hydraulic radii are calculated by MOUSE.

### MIKE SHE Overland flow to MOUSE LINKS

If the MOUSE links are defined as “open” channels, then MIKE SHE can discharge overland flow directly into the MOUSE links. In this case, the exchange coefficient in Equation (11.1) is defined as

$$C = C_L \cdot L \quad (11.7)$$

where  $C_L$  is the conductance and  $L$  is the length of the MOUSE pipe (link) in the MIKE SHE cell.

If the exponent Equation (11.1) is 1.0, then this is a simple drain formulation and the conductance is per length with units of [m/s]. If the exponent is 1.5, then this is a weir formulation and the units of the conductance term are [m<sup>1/2</sup>/s].



### MIKE SHE Overland flow to MOUSE Manholes

If the MOUSE manholes are not sealed, then MIKE SHE can discharge overland flow into the MOUSE manholes. In this case, the exchange coefficient in Equation (11.1) is defined as

$$C = C_L \quad (11.8)$$

where  $C_L$  is the conductance.

If the exponent Equation (11.1) is 1.0, then this is a simple drain formulation and the conductance,  $C_L$ , is per length with units of [m/s]. If the exponent is 1.5, then this is a weir formulation and the units of the conductance term are [m<sup>1/2</sup>/s].

### MIKE SHE SZ drain flow to MOUSE Manholes

If drain flow is specified in MIKE SHE, then the drainage can be discharged to a MOUSE manhole. The flow in the drain is calculated by MIKE SHE based on the groundwater height above the drain level. In MIKE SHE the distributed drainage option must be chosen (see Drainage (V.2 p. 123)) and the cells that drain to a manhole must have an option value of 4 (see Option Distribution (V.2 p. 128)). The references between the MIKE SHE drain codes and the MOUSE manholes are defined in the *MsheMouse.pfs* file (see Creating a MsheMouse.pfs file (V.1 p. 184)).

### MIKE SHE Paved Areas to MOUSE Manholes

If the paved area option (see Land Use (V.2 p. 62)) is used in MIKE SHE, then the flow generated on the paved areas can be discharged to a MOUSE manhole. MIKE SHE's paved area flow module uses the same reference system as the drain component. This option is automatically activated when the MIKE SHE drains in the paved areas point to a MOUSE manhole.

## 11.1 Coupling MIKE SHE and MOUSE

The MOUSE coupling in MIKE SHE has not yet been added to the MIKE SHE user interface. Thus, to couple the models together, you must:

- 1 tell MIKE SHE to look for a MOUSE model,
- 2 tell MOUSE that it is coupled to a MIKE SHE model
- 3 create an *MsheMouse.pfs* file to define where and how the two models are coupled.



### 11.1.1 Telling MIKE SHE to couple to MOUSE

To tell MIKE SHE that it needs to couple to a MOUSE model, you must add the following two items in the Extra Parameters section of the MIKE SHE Setup Editor.

| Parameter Name      | Type      | Value   |
|---------------------|-----------|---|
| mouse coupling      | Boolean   | On  |
| mouse coupling file | file name | the file name of the MOUSE coupling .pfs input file |

Note, that the parameter names must be spelled exactly as shown. For more information on the use of extra parameters see Extra Parameters (V.1 p. 143).

### 11.1.2 Telling MOUSE that it is coupled to a MIKE SHE model

To couple a MOUSE model to MIKE SHE, MOUSE must be supplied with some extra information. This information is found in MOUSE's .ADP file.

| Line item  | Comment  |
|--|--|
| [MOUSE_COUPLING]<br>SYNTAX_VERSION = 1<br>UNIT_TYPE = 1<br>CALLER = 'MSHE'         |  |
| // LineHeader = 'ID', 'LinkType', 'C ', 'OLExp',<br>'SzLeakageCoef'                | Comment line for headers   |
| COUPLINGMMSHE= 'NODE1', 1, 0.001, 2, ,<br>COUPLINGMMSHE= 'LINK1', 2, 0.001, 2, 0.2 | One line for each coupling item:<br>ID = Link name<br>LinkType = 1 for node; 2 for link<br>C = conductance for Overland flow to MOUSE, units depend on OLExp and whether it is a pipe or a manhole<br>SzLeakageCoeff = leakage coefficient; needed only when the saturated zone is coupled to a link |
| [Endsect]  |  |



### 11.1.3 Creating a *MsheMouse.pfs* file

The *MsheMouse.pfs* file is an ASCII file that includes all of the specifications for the coupling. The following table defines the structure of the file, along with some information on the parameters. When the MOUSE coupling has been added to the user interface, the creation of this file will be automatic.

Table 11.1 *MsheMouse.pfs* file format and description

| Line item  | Comment   |
|--|---|
| [MIKESHE_MOUSE_Specifications]<br>FileVersion = 2        |   |
| Link_SZ_Exchange_Option = 2                              | 1 = Leakage coefficient based only on<br>MOUSE pipe leakage coefficient<br>2 = Leakage coefficient based on a<br>series connection of the MOUSE pipe<br>leakage coefficient and the MIKE<br>SHE aquifer properties  |
| Mouse_MPR_file name =<br> . \MOUSE_NASSJO\handskeryd.mpr | Name of the mpr file. The MIKEZero<br>file name format (     ) indicates that<br>the file name is relative to the location<br>of this document.   |
| SZ_Coupling = 1  | 1 or 0 to include/exclude SZ<-<br>>MOUSE coupling   |
| OL_Coupling = 1  | 1 or 0 to include/exclude Overland<-<br>>MOUSE coupling   |
| Dynamic_Coupling = 1                                     | 1 for dynamic coupling. Otherwise the<br>initial MOUSE conditions will be<br>used.  |
| Drainage_To_Manholes = 1                                 | 1 to include SZ (and paved area) drain<br>to manholes. In this case the SZ drain<br>option must be Levels and Codes<br>(should rather be named Distributed<br>Option). In the areas with drain to<br>MOUSE the Distributed option code<br>must be 4. For each drain code value<br>found in areas with Distributed code 4<br>a reference from the code to a<br>MOUSE manhole must be defined in<br>the Drainage_Manholes section (see<br>below). |



Table 11.1 *MsheMouse.pfs file format and description*

| Line item  | Comment  |
|--|--|
| Smooth_SZ_Inflow = 1<br>Smooth_OL_Inflow = 1   | Ensures a more smooth calculation of flows to MOUSE when the MIKE SHE time steps are large compared to the MOUSE time step. The MOUSE coupling is only made at every integer multiple of the MIKE SHE time step. If the Smooth option is not activated, the flows to MOUSE can stop after a number of MOUSE time steps because the calculated flow volume exceeds the volume of the MIKE SHE SZ/Overland grid cells. The Smooth option tries to use a reduced flow rate which equals the available volume / coupling time. |
| [Dynamic_Coupling_Specifications]  |  |
| Limit_Inflow = 0:  | Specify 1 if the inflow to MOUSE should be limited so the MOUSE volume + inflow does not exceed a specified fraction of the maximum MOUSE volume. This is used to avoid instabilities due to high pressure.  |
| Limit_Outflow = 0:   | Specify 1 if the outflow from MOUSE should be limited so the MOUSE volume - outflow doesn't come below a specified fraction of the maximum MOUSE volume. This is used to avoid instabilities due to drying / negative volume.  |
| [Inflow_Limitations]<br>MaxVolFac_Links = 0.99<br>MaxVolFac_Manholes = 0.99<br>EndSect // Inflow_Limitations<br>[Outflow_Limitations]<br>MinVolFac_Links = 0.05<br>MinVolFac_Manholes = 0.05<br>EndSect // Outflow_Limitations | The inflow and outflow fractions are specified here:   |
| EndSect // Dynamic_Coupling_Specifications   |  |



Table 11.1 MsheMouse.pfs file format and description

| Line item  | Comment   |
|--|---|
| No_Of_Storing_reaches = 2<br>[Storing_Reaches]<br>[Storing_Reach_1]<br>No_Of_Links = 2<br>LinkName_1 = 'Dike_0111'<br>LinkName_2 = 'Dike_0311'<br>EndSect // Storing_Reach_1<br>[Storing_Reach_2]<br>No_Of_Links = 1<br>LinkName_1 = 'Dike_0411'<br>EndSect // Storing_Reach_2<br>EndSect // Storing_Reaches | When No_Of_Storing_reaches is greater than 0, the [Storing_Reaches] section must be specified, and inside this the [Storing_Reach_1], [Storing_Reach_2], ... defining the no. of links and link names for each reach. |
| [Drainage_Manholes]<br>No_Of_DrainCodes = 8<br>[Draincode_1]<br>Draincode= 12<br>ManholeName='DNB3182'<br>Endsect // Draincode_1<br>.<br>.<br>Endsect // Draincode_8<br>EndSect // Drainage_Manholes   | When No_Of_Storing_reaches is greater than 0, the [Storing_Reaches] section must be specified, and inside this the [Storing_Reach_1], [Storing_Reach_2], ... defining the no. of links and link names for each reach. |
| EndSect // MIKESHE_MOUSE_Specifications  |   |

#### 11.1.4 Output Files

Output from the coupled run is written to a number of .dfs0 results files—all located in the standard results directory. In the case of storing reaches, there is one item in the .dfs0 file for each storing reach.

Table 11.2 File names and conditions for output for the MIKE SHE-MOUSE coupling. 'setupname' refers to the name of the model setup file.

| file name                                   | The file is created when...                                     |
|---|---|
| .\setupname\setupname_SZ2MouseReaches.dfs0  | ...the MIKE SHE SZ coupling is included.                        |
| .\setupname\setupname_OL2MouseReaches.dfs0  | ...the MIKE SHE Overland coupling is included.                  |
| .\setupname\setupname_OL2MouseManholes.dfs0 | ...the MIKE SHE Overland flow coupling to manholes is included. |



Table 11.2 File names and conditions for output for the MIKE SHE-MOUSE coupling. 'setupname' refers to the name of the model setup file.

| file name   | The file is created when...                                |
|---|--|
| .\setupname\setupname_SZDrain2MouseManholes.dfs0    | ...the MIKE SHE SZ drain coupling to manholes is included. |
| .\setupname\setupname_PavedDrain2MouseManholes.dfs0 | ...the MIKE SHE SZ paved areas to manholes is included.    |

## 11.2 Running the Coupled Models

The execution is controlled by MIKE SHE. The MShe\_PreProcessor and MShe\_WaterMovement call functions in the MOUSE\_HD.dll. The present version requires that the mouse\_hd.dll and other mouse files are located together with the MIKE SHE .exe files in the MIKEZero\bin directory.

Thus, before running the coupling for the first time, you should run the following small .bat file that copies all of these files from the MOUSE install directory to the MIKE SHE install directory:

```
MZero\bin\copyMOUSEFilesToMZ.bat
```

Before running this file, you should open the .bat file and check that your install directories are not different than those listed in the .bat file.

### 11.2.1 Warning messages

#### Exchange inflows reduced

Warning: Exchange inflows from Overland to MOUSE reduced by Overland house-keeping in order to avoid instabilities

No. of time steps: 27000 of 27000

Total a priori inflows: 1332286 m3

Total reduced inflows: 920643.0 m3 (69.10%)

MIKE SHE calculates time in/out flows after an overland time step and feeds them to MOUSE for one or more MOUSE time steps. The calculations of these flows are not included in the implicit overland flow solver. Thus, the "Total a priori flows" are the rough inflows calculated using Equation (11.1). However, to prevent water balance errors, MIKE SHE checks the volume of water available in the grid cell. If the volume is insufficient, then the inflow is reduced to the available amount. The final value of inflows is the "Total reduced inflows". Note though that the total



NET inflow to MOUSE will be less than this value if the flow goes from MOUSE to MIKE SHE in other grid cells or other time steps.

Ideally, the Total reduced inflow should be 100%, but in practice this is rarely achieved.

### **11.2.2 Water Balance Limitation**

The interaction with MIKE SHE is not included in the MOUSE Summary HTM file. Thus, the water added from MIKE SHE appears as an error (i.e. 6: Continuity balance).



*PHI Software*

**WORKING WITH WATER QUALITY**





## 12 **SIMULATING WATER QUALITY**

The complete MIKE SHE advection-dispersion (AD) module is comprised of four components, each describing the transport processes in one of the parts of the hydrological cycle. Used in combination they describe solute transport in the entire hydrological cycle. The four components are:

- Overland Transport
- Channel Transport (MIKE 11)
- Unsaturated Zone Transport
- Groundwater Transport

In principle, these processes are independent. However, the current version of MIKE SHE AD can only calculate the solute transport

- for all components together (overland, unsaturated zone and groundwater ),
- for the saturated zone and the overland component together, or
- for the saturated zone alone.

A number of processes relevant for simulating reactive solute transport are included in MIKE SHE including

- Water and solute transport in macro pores,
- Sorption of solutes described by either equilibrium sorption isotherms (Linear, Freundlich or Langmuir) or kinetic sorption isotherms, which include effects of hysteresis in the sorption process,
- Attenuation of solutes described by an exponential decay, and
- Plant uptake of solutes.

### **2007 Release versus 2005 Release**

In the 2007 Release, the saturated zone and overland components of the AD module are included in the user interface. The user interface does not yet include the UZ component or the water quality processes for overland, unsaturated and saturated flow. The particle tracking (PT) module is also not yet supported in the user interface.

In the mean time, the full integrated functionality of the AD module, including all the water quality processes and the PT module, is available through the command line interface in the same manner as it was available in the 2005 Release. This is described in *Using the Fully Integrated AD Module (V.1 p. 195)* and *Working with Particle Tracking (V.1 p. 227)*



## 12.1 Flow Storing Requirements

Solute transport calculations in MIKE SHE AD are based on the water fluxes from a MIKE SHE Water Movement (WM) simulation. To ensure that all the needed WM result data types are stored, you have to specify that results should be stored for an AD simulation. See *Storing of Results (V.2 p. 135)*.

The WM data should be stored frequently enough to describe the dynamics of the flow. The selected storing frequencies of flow results will usually be a compromise between limitations in disk space and resolution of the flow dynamics. The maximum computational time steps in a transport simulation are often restricted by advective and dispersive stability criteria. However, the transport time step cannot be greater than the flow storing time step in each component.

## 12.2 Storing of Results

The simulated concentration distribution in each component as well as the mass balances and fluxes will be stored in dfs2 and dfs3 files with different time steps. Besides these result files, the program also writes output to the error log, which describes errors encountered during execution and a print log which contains execution step information, statistics on the run and a mass balance (if requested).

Normally, the results from the saturated zone (species concentration in each grid) is by far the most disk consuming parameter. So, be careful with the storing time step. Mass balances, which includes time series of mass storage and fluxes between components (and sources, drains, boundaries etc.) can be stored at smaller time steps.

When you select the time step you should also be aware of the time scale of the process. The time scale for transport processes in groundwater is usually much larger than the time scale for transport in a river.

Enter the desired time steps - notice that the unit is hours - in each of the edit fields. There are no limitations on this time step but if you select a time step less than the simulation time step, the storing time step will be the new simulation time step.





## 12.3 Simulation and Time Step Control

Simulation time steps are to some extent controlled by the user. Several possibilities for time step control exist to make the execution as fast as possible with no numerical dispersion and instabilities.

The first possibility for controlling the simulation time steps in the different components is simply to define the maximum time step in each component. Note that time steps should be given in increasing order i.e.  $dt_{RIVER} \leq dt_{OVERLAND} \leq dt_{UZ} \leq dt_{SZ}$ . Also note that this is the MAXIMUM time step. That is, the actual simulation time step is controlled by the stability criterions with respect to advective and dispersive transport given below. Furthermore, time steps for transport cannot exceed the storing time step for the relevant data in the flow result file from a MIKE SHE flow simulation.

Enter the maximum allowable Courant number for each component. The Courant number is defined by  $V \times dt/dx$  (velocity times time step divided by “grid size”). This number should normally not exceed 1.0 for one and two-dimensional transport (UZ, OVERLAND and RIVER) and 0.8 for three-dimensional transport (SZ). The maximum time step will be calculated accordingly.

Enter the maximum allowable dispersive Courant number for each component. The dispersive Courant number is defined by  $D \times dt/dx^2$  (Dispersion coefficient times time step divided by “grid size” squared). This number should normally not exceed 0.5. The maximum time step will be calculated accordingly.

The transport limits are used to avoid negative concentrations in cases with extreme gradients (e.g. close to sources) or in areas with highly irregular velocity fields. Enter the maximum allowable transport from a node or grid as a fraction of the storage in the node or grid. A recommended value for all components is 0.9, which ensures that this option is in use (the value 0 determines that this option is not in use).

### 12.3.1 Calibrating and Verifying the Model

The advection-dispersion of solutes depends largely on the simulated flows and fluxes calculated by the MIKE SHE flow model. After your first AD simulations, you will usually have to go back and improve the calibration of your flow model. Rarely, can the simulated concentrations and mass fluxes be calibrated to the measured concentrations by tuning only the solute transport model.



It is important to recognise that a transport model must be calibrated. This is true for all applications larger than the laboratory scale since model output cannot necessarily be compared directly to measured values. Measurements are mostly point measurements at a certain time whereas results often are mean values over larger volumes and longer times.

The purpose of the calibration is to tune the model so that it is able to reproduce measured conditions for a particular period in a satisfactory way. This period - known as the calibration period - should be chosen long enough to include events of similar kind as the ones you are going to investigate.

A satisfactory calibration is reached when the model is able to reproduce the measured values taking the following conditions into account:

- uncertainty in the measurements (time, space, equipment)
- representativeness of measurements (point/average grid values)
- differences between your conceptual model and nature
- uncertainty in other model parameters and data (source description etc.)

In general, it is impossible to specify an exact level of divergence between measured data and computed results before the model is satisfactorily calibrated. In each application you have to consider all factors influencing your result.

After the calibration, you should verify your model by running one or more simulations for which measurements are available without changing your model parameters. If the model is able to reproduce the validation measurements you can consider your calibration to be successful. This ensures that simulations can be made for any period similar to the calibration and the verification period with satisfactory results.



## 13 USING THE FULLY INTEGRATED AD MODULE

If you only want to calculate water quality in the saturated zone or in the overland flow zone, then you do not need to use the command line and file interface described in this chapter. This chapter is meant for those users who want to calculate water quality in the unsaturated zone or to calculate sorption and degradation. When the fully integrated AD user interface is available, this chapter will no longer be necessary.

Working with the advection-dispersion module involves handling of a number of text files with normal text editors. The input data files for a simulation with the AD module are called the transport set-up files - .tsf and .xtsf. These are both text files, which contain all relevant information for the simulation. They must be located in the project's home directory. It is essential for a successful simulation with the advection-dispersion module that the formats described below are followed.

Two-dimensional input data as well as time dependent input data are given in the usual MIKE ZERO formats for such data i.e. dfs2 and dfs0 formats. All results are also stored as dfs2 data and are ready to be handled by the different editing and presentation tools.

### 13.1 Data Units

All units for **constants** related to distances, areas and volumes should be given in **metric** units (m, m<sup>2</sup>, and m<sup>3</sup>). All units for constants related to time should be given in **seconds**. Internally, the AD module is using *g* for mass and *g/m<sup>3</sup>* for concentration.

#### Units for Constants must be Base Units

For data that can be read as either a constant or from a dfs file, the unit for constant values depends on the Base Unit for the data type. For the dfs2/dfs0 files you can select the most suitable unit. The Base Unit can be found by looking at the available units for a data type. The first allowed unit for the data type is the Base Unit.

### 13.2 Executing MIKE SHE AD

The fully integrated 2005 version of MIKE SHE AD is executed from a DOS prompt. Open a DOS window and navigate to the directory where the transport set-up file is located. The simulation is started by the command:



```
MShe_AdvectionDispersion projectname.tsf \\apv
```

A window similar to the Water Movement simulation showing the simulation status will appear.

**Note** The AD executable cannot be executed from the MZLaunch utility because it still uses the 2005 \\apv command.

MIKE SHE AD can also be executed in batch mode simply by writing the above command in a .bat file and executing this file by either doubling clicking on it or typing the name in a DOS prompt. This allows you to run successive simulations of various models.

MIKE SHE AD may also be executed together with the AUTOCAL routine allowing you to do automatic parameter optimization. See the MIKE Zero AUTOCAL user guide for details and just exchange the .fsf file with the .tsf file in the description.

### 13.3 Working with the .TSF Files

The input data file for a simulation with the AD module is called the transport set-up file - tsf. This is a text file, which contains all relevant information for the simulation and it should be located in the project home directory.

The tsf file is basically the same file as the one used in previous versions of MIKE SHE with two major changes:

- results are now read from dfs2 and dfs3 files produced by the latest version of MIKE SHE.
- As the .dfs2 and .dfs0 files can contain multiple items, you must specify the item number along with the file name.

The tsf file is divided into four parts:

- simulation parameters
- species independent parameters
- species dependent parameters
- extra input “the bottom of the tsf file”

In the next sections, each of these input parts is explained and illustrated in detail.



The output of the MIKE SHE AD is now stored to a number of dfs2 and dfs3 files which can be viewed and processed with the different tools available for these files in MIKE ZERO.

*Table 13.1 MIKE SHE AD .tsf file format and description*

| Line item  | Comment  |
|--|--|
| <pre> FILETYPE DATA TYPE VERN0: 2001 1      524 ===== SETUP DATA for Transport Simulation ===== THIS FILE OK (T/F) : T ===== </pre>  | <p>The tsf file starts with a header giving the information on the type and version of the input file. The current version should be 524</p> <p>The second line is for information only.</p> <p>The file structure was originally conceived for an X-Windows interface and therefore contains a check on the integrity, which was used by the GUI. These lines should be left unchanged:</p> |
| <pre> RESULTS OF WATERMOVEMENT CALCULATION file name : projectname\projectname.frf ===== </pre>  | <p>The MIKE SHE AD simulation will be based on the flow field, heads, water depths, etc. calculated by a MIKE SHE WM calculation.</p>  |
| <pre> SPECIES IN CALCULATION No. of species      : 1 ===== </pre>  | <p>MIKE SHE AD allows for calculating multi-species transport. In the next lines you specify the number of solute species that will be used. For each of these species input will be required in the species dependent input section.</p>  |
| <pre> SIMULATION CONTROL PARAMETERS ----- Simulation Identification Simulation title   : example of MIKE SHE AD input file Simulation descr   : integrated model for solute transport </pre> | <p>A title and some text describing the simulation. These titles will appear in the log files that are written during the calculation.</p>   |



Table 13.1 MIKE SHE AD .tsf file format and description

| Line item   | Comment   |
|---|---|
| <p>----- Simulation period</p> <p>Cycle Option 1/2/3 : 2</p> <p>Start year : 1994</p> <p>Start month : 1</p> <p>Start day : 2</p> <p>Start hour : 0</p> <p>Start minute : 0</p> <p>End year : 2001</p> <p>End month : 12</p> <p>End day : 31</p> <p>End hour : 0</p> <p>End minute : 0</p> <p>Cycle restart year : 1994</p> <p>Cycle restart month: 1</p> <p>Cycle restart day : 1</p> <p>Cycle restart hour : 0</p> <p>Cycle restart min : 0</p> <p>Cycle end year : 1997</p> <p>Cycle end month : 12</p> <p>Cycle end day : 31</p> <p>Cycle end hour : 0</p> <p>Cycle end minute : 0</p> <p>Stat. Flow year :</p> <p>Stat. Flow month :</p> <p>Stat. Flow day :</p> <p>Stat. Flow hour :</p> <p>Stat. Flow minute :</p> | <p>The solute transport simulation period should be a part of the flow simulation period. However, either a saved WM time step or a time period from the WM simulation can be recycled over the AD simulation period.</p> <p>The choice for how the WM time period relates to the AD time period is defined by the Cycle Option. Valid values are 1,2, and 3.</p> <p>1 no recycling: The AD will be run from the start date to the end date, which both must be in the WM simulation period.</p> <p>2 recycling: The AD will be run from the start date to the end date, but the WM results will start at the Cycle restart date and be used until the Cycle end date is reached, at which point the WM results will start to repeat.</p> <p>3 constant flow field: The AD will be run from the start date to the end date, using a stationary WM saved time step defined by the Stat. date.</p> <p>In the example to the left, the AD simulation will run from 2 Jan 1994 00:00 until 31 Dec 2001 00:00. The WM results will be used starting at 1 Jan 1994 00:00, until 31 Dec 1997 00:00, at which time the WM results will be repeated starting at 1 Jan 1994 00:00.</p> <p>Note that since the stationary flow field option not used, a Stat. Flow date is not required.</p> |
| <p>----- Execution Logicals</p> <p>Execute SZ T/F : F</p> <p>Execute All. T/F : T</p> <p>Execute OC T/F :</p>   | <p>In the present version of MIKE SHE AD</p> <ul style="list-style-type: none"> <li>the groundwater transport (SZ) can run by itself,</li> <li>the groundwater transport can run in combination with the overland (OC) transport, or</li> <li>transport can be calculate for all the hydrologic components - overland, unsaturated, and saturated flow.</li> </ul> <p>Thus, single combinations with unsaturated flow are not possible.</p> <p>Note, that you cannot calculate transport unless the WM simulation has included this component.</p>  |
| <p>----- Extensions</p> <p>SZ Double Por. : F</p>   | <p>T: Diffusion processes into immobile fractures in the saturated zone will be included.</p> <p>F: These processes will not be included.</p>   |
| <p>----- Inclusions of sources</p> <p>SZ T/F : T</p> <p>UZ T/F : T</p> <p>Overland T/F : T</p> <p>River T/F : T</p> <p>Prec./Inf. T/F : T</p>   | <p>Here, you specify which types of sources will be included.</p> <p>In the Species Independent section, you specify the location of the sources and in the Species Dependent section, you specify the source strengths.</p>  |



Table 13.1 MIKE SHE AD .tsf file format and description

| Line item   | Comment   |
|---|---|
| <pre> ----- Storing of Results Storing SZ          T/F: T Storing UZ          T/F: T Storing OverlandT/F: T Storing River      T/F: F Storing MBL r      T/F: T St.Frq  SZ          : 2160.000000 St.Frq  UZ          : 2160.000000 St.Frq  Overland    : 2160.000000 St.Frq  River       : 2160.000000 St.Frq  MassBal.    : 2160.000000 Print frq MBL       : 8640.000000 </pre>  | <p>The storing of results is defined per component, not by species (MBL = mass balance)</p> <p>Thus, all results will be stored for all species for each component set to (T) rue - at the specified storing frequency in hours.</p> <p>Note. The River component must be included, but is no longer active, as it is calculated by MIKE 11.</p>  |
| <pre> ----- Time Step Control Max SimTimeStep SZ : 100000000000000.00 Max SimTimeStep UZ : 100000000000000.00 Max SimTimeStep Ovl: 100000000000000.00 Max SimTimeStep Riv: 100000000000000.00 Max Advec.Cour. SZ : 0.800000 Max Advec.Cour. UZ : 0.800000 Max Advec.Cour.Ovl : 0.800000 Max Advec.Cour.Riv : 0.800000 Max Disp.Cour. SZ : 0.600000 Max Disp.Cour. UZ : 0.600000 Max Disp.Cour. Ovl : 0.600000 Max Disp.Cour.Riv : 0.600000 Max Transport SZ   : 0.900000 Max Transport UZ   : 0.900000 Max Transport Overl: 0.900000 Max Transport River: 0.900000 ===== </pre> | <p>The time step control in AD consists of 6 elements:</p> <p><b>WM result storing time step:</b> the AD time step may not exceed the WM timestep;</p> <p><b>AD storing time step:</b> the AD time step may not go past an AD storing time step time;</p> <p><b>source/sink input:</b> the AD time step may not exceed the time for which source data needs to be read;</p> <p><b>Max SimTimeStep:</b> the maximum AD time step;</p> <p><b>Max Advec/Disp Cour:</b> the AD time step must be such that the advective and dispersive courant criteria are not exceeded;</p> <p><b>Max Transport:</b> the AD time step must be such that the mass transport between cells does not exceed this specified fraction.</p> <p>In practice the max time step is often set to a very high value to ensure that the time step is controlled by the other criteria.</p> |

### 13.3.1 Species INDEPENDENT Data

Compared to earlier versions the input is now given as dfs2 and dsf0 files, which replace the old T2 and T0 file formats. In the input the user now has to specify **BOTH a file name and an item number**. As a dfs file can possibly contain multiple items the program needs the item number to identify which dfs item it needs to read from the file. The dfs0 and df2 files can be prepared with the tools in the MIKE ZERO user interface. The items in the dfs files have to have the right data types to be used by MIKE SHE AD. These are listed in Table 13.2.



The units can be set in the dfs file. In case a constant value is specified this value has the base unit

*Table 13.2 Input data types, their correct EUM data types and the base unit which is used when specifying constant values*

| Input Data                        | EUM autotype in dfs file   | Base unit for constant values |
|-----------------------------------|----------------------------|-------------------------------|
| effective porosity                | Porosity Coefficient       | fraction between 0 and 1      |
| matrix porosity                   | Porosity Coefficient       | fraction between 0 and 1      |
| diffusivity (SZ, UZ)              | Dispersion Velocity Factor | [m]                           |
| source location                   | Grid Codes                 | [integer codes]               |
| diffusion coefficient (OC, River) | Dispersion coefficient     | [m <sup>2</sup> /s]           |

Input for layers is given in the format:

```
No of data elements      :      2
Lower layer              :      1
Parameter                :      MAPS/File.dfs2 2
Lower Layer              :      99
Parameter                :      0.01
```

This should be interpreted as if there are two sets of data input where each set is for a particular layer interval. The first set applies from layer 1 to the Lower layer for the first set. The next one will apply from the lower layer of the previous set + 1 until the Lower layer specified. The data sets must at least cover all layers i.e. the user must specify a value equal or larger than the number of layers in the model as the last Lower layer (but a larger





value is allowed). The actual parameter input can be either a constant value or a dfs2 input file. .

Table 13.3 MIKE SHE AD .tsf file format and description continued

| Line item   | Comment  |
|---|--|
| <pre>===== SZ INPUT PART ----- Effective porosity No of data elements: 2 Lower layer      : 1 Porosity         : MAPS\Porosity.dfs2 Lower layer      : 99 Porosity         : 0.01</pre> | <p>Fluxes of water are automatically read from a flow result file according to the storage frequency in the specified simulation period. Together with these fluxes the effective porosity in the groundwater determines the advective velocity of the species. The effective porosity is always in the range between 0 and 1 i.e. for porous media usually 0.15 to 0.3 depending of the grain size distribution (the more uniform the higher effective porosity) and for fractured media usually 0.01 to 0.05.</p> <p>The effective porosity can be given either as a uniform value or as a fully distributed number. In the first case you should just specify the “bottom” layer in you set-up as 'Lower layer' and the value in the edit field 'Porosity'. In the latter case you can specify a dfs2 matrix data file with the distributed porosity for each of the layers in your set-up.</p> |
| <pre>-----Type of dispersion description Aniso. opt. 1/2/3 : 1</pre>  | <p>For the dispersion parameters two different options have been included in MIKE SHE AD i.e. isotropy and anisotropy with axial symmetry around the z-axis. These simplifications are identified on the number of non-zero dispersivities to be specified. Under the assumption of isotropic conditions you should only specify the longitudinal dispersivity, <math>\alpha_L</math>, and the transversal dispersivity, <math>\alpha_T</math> - the other three dispersivities are automatically set. Under anisotropic conditions you should specify five dispersivities.</p> <p>For the “Aniso. opt.” the user can set the following:</p> <ul style="list-style-type: none"> <li>1 no dispersion</li> <li>2 isotropic</li> <li>3 anisotropic</li> </ul>   |



Table 13.3 MIKE SHE AD .tsf file format and description continued

| Line item   | Comment   |
|---|---|
| <p>----- Dispersivities</p> <p>No of data elements: 0</p>   | <p>The sizes of these parameters depend on the degree of heterogeneity in your geology and other matters that affects the velocity field <b>and</b> the degree to which these matters have been described in the flow description. The more heterogeneous your geology is the larger dispersivities should be applied and the more detailed you have described the heterogeneities the smaller dispersivities should be applied.</p> <p>Furthermore, the size of the dispersivities depend on the scale of modelling <b>and</b> on the applied grid size. The larger scale the larger dispersivities and the larger grid size the smaller dispersivities should be applied because of numerical dispersion.</p> <p>For these reasons it is difficult to give rule of thumb values for the dispersivities. Recent field experiments on solute transport though indicate that the longitudinal dispersivity should be in the range of 1% or less of the travel distance, the transversal, horizontal dispersivity should be at least 50 times less than this and the transversal, vertical dispersivity should be even 2 or more times less again. Dispersivities are specified in the unit metres. In the example below we assume there is no dispersion (option 1) and no values are needed</p> |
| <p>----- Dispersivities</p> <p>No of data elements: 1</p> <p>Lower Layer : 99</p> <p>Disp. alfa LHH : MAPS\SZDispersivity.dfs2 1</p> <p>Disp. alfa THH : 0.05</p> <p>Disp. alfa TVH :</p> <p>Disp. alfa LVV :</p> <p>Disp. alfa THV :</p> | <p>In case the option 2 (isotropic) is chosen the user has to specify the horizontal, longitudinal and transversal dispersivity. As shown below input for the three other components is not needed as these are only used when using option 3. The values can be given as a constant value or as a map using dfs2 files.</p>  |



Table 13.3 MIKE SHE AD .tsf file format and description continued

| Line item  | Comment  |
|--|--|
| <p>If dual porosity is not used:</p> <pre>----- Matrix porosity No of data elements: 0</pre> <p>or if dual porosity is used:</p> <pre>----- Matrix porosity No of data elements: 1 Lower Layer          : 99 SZ matrix porosity  : MAPS\Matrixporosity.dfs2 1</pre>  | <p>Solutes in a fractured media will be transported by diffusion in and out of the soil matrix of the media causing very fast breakthroughs and long tailings. This process can be included in MIKE SHE AD by activating the dual porosity transport component in which case you should specify the matrix porosity and mass transfer coefficient of the medium. Matrix porosity is given as a value between 0 and 1 and as for effective porosity you can either specify a uniform value for the entire area or distributed values using dfs2 files.</p> <p>Matrix porosities are generally very difficult to measure and application of this component requires often calibration against breakthrough curves to give realistic estimates of the parameters. Furthermore, input should be the “effective” matrix porosity i.e. the matrix porosity that is “actively” involved in the solute diffusion. This can be significantly lower than the matrix porosity measured by core analysis. For a limestone aquifer the matrix porosity has been calibrated to be as small as 4 per cent (core samples indicated 20 to 35%) whereas for a clay till sample it was calibrated to be 20 to 30% - a little less than the total matrix porosity. In our example the dual porosity option is not used and no input is required.</p> |
| <pre>----- Source locations No of data elements: 2 Location number    : 1 Upper Layer        : 1 Lower Layer        : 2 Source type        : 3 Area distribution   : X coordinate        : 9500 Y coordinate        : 17500 Location number    : 2 Upper Layer        : 1 Lower Layer        : 1 Source type        : 4 Area distribution   : MAPS\SZSourceLocation.DFS2 1 X coordinate        : Y coordinate        : =====</pre> | <p>Mass of solutes can externally be introduced into the groundwater transport component in four different ways i.e. as a point or line (over depth) source in specific grids or as a spatially distributed source in a certain depth interval. In both cases the source can either be time varying flux of mass (mass/time [point or line] or mass/area/time [area source]) or fixed concentrations (mass/volume). Observe that fixed concentrations also may vary in time: Table 13.4 presents the possible combinations:</p> <p>A point or line source is introduced by specifying the upper and lower layer and the X and Y co-ordinates of the horizontal location of the point (“grid”) in the model co-ordinate system. A spatially distributed source is introduced by specifying the upper and lower layers and the spatial distribution as a dfs2 file with code '1' in the source area and '0' elsewhere.</p> <p>The source strength is given in the Species Dependent input part.</p>  |



Table 13.3 MIKE SHE AD .tsf file format and description continued

| Line item  | Comment   |
|--|---|
| <p>UZ INPUT PART</p> <p>----- Dispersivities</p> <p>No of data elements: 2</p> <p>Depth : 1</p> <p>Dispersion : MAPS\UZDisper-</p> <p>sion.dfs2 2</p> <p>Depth : 99.9</p> <p>Dispersion : 0.0</p>  | <p>For UZ, which is 1D, the dispersivity is specified as a single dispersivity value. Each of the input elements consists of a depth input indicating the depth in meters below ground surface to which the dispersion input is valid and the actual value to use which can be either a constant value or a dfs2 file.</p> <p>The same comments as given for dispersion in groundwater apply for solute transport in unsaturated media. The few studies reported in the literature on solute transport in unsaturated porous media suggest dispersivities around 0.1 metre for travel distances less than 2 metres.</p> <p>The longitudinal dispersivity can be distributed over depth by specifying depth intervals (as described above). Depth and dispersivity are specified in metres.</p>  |
| <p>----- Source locations</p> <p>No of data elements: 2</p> <p>Location number : 1</p> <p>Upper depth : 0.5</p> <p>Lower depth : 1.0</p> <p>Source type : 1</p> <p>Area distribution : X coordinate</p> <p>: 18600</p> <p>Y coordinate : 11700</p> <p>Location number : 2</p> <p>Upper depth : 0.5</p> <p>Lower depth : 1</p> <p>Source type : 2</p> <p>Area distribution : MAPS\UZSourceLoca-</p> <p>tion.dfs2 1</p> <p>X coordinate :</p> <p>Y coordinate :</p> <p>=====</p> | <p>Normally, solutes are introduced in the unsaturated zone by the precipitation, and MIKE SHE determines the infiltration rate and thereby the mass flux in the upper node. However, mass of solutes can externally be introduced into the unsaturated zone transport component in two other ways namely as a point or line source over a certain depth in a specific soil column (grid) or as spatially distributed source in a certain depth interval. In both cases the source is given as time varying flux of mass (mass/time or mass/area/time).</p> <p>A point or line source is introduced by specifying the upper and lower layer and the X and Y co-ordinates of the horizontal location of the point ("grid") in the model co-ordinate system. A spatially distributed source is introduced by specifying the upper and lower depth and the spatial distribution as a dfs2 file with code '1' in the source area and '0' elsewhere. Input that varies with depth can be given in UZ over depth intervals i.e. the user specifies the depths (depth1, depth2,..., depthN as numbers) and the parameter distributions in the entire model area for that depth interval as a dfs2 data file or a constant value. The parameters will then be uniform in each grid from soil surface to depth1 from depth1 to depth2 etc. until the bottom of the unsaturated zone is reached. This way of giving of values for the unsaturated zone transport component has the advantage that the user does not have to worry about the vertical discretisation. But has the disadvantage that it does not take into account the discretisation which can vary from one UZ column to the next.</p> <p>Depth should be given as meters below ground surface. Source strengths are specified in the Species Dependent input part.</p> |



Table 13.3 MIKE SHE AD .tsf file format and description continued

| Line item  | Comment  |
|--|--|
| OVERLAND INPUT PART<br>----- Dispersion<br>X-disp. coeff. : 0.0<br>Y-disp. coeff. : 0.0  | For the 2D overland transport component two dispersion coefficients are specified. Notice that the dispersion coefficient ( $\text{m}^2/\text{s}$ ) is specified directly and not a dispersivity ( $\text{m}$ ) as was the case for the SZ and UZ.   |
| ----- Source locations<br>No of data elements: 2<br>Location number : 1<br>Source type : 1<br>Area distribution :<br>X coordinate : 19500<br>Y coordinate : 17500<br>Location number : 2<br>Source type : 2<br>Area distribution : MAPS\OLSrc.dfs2 1<br>X coordinate :<br>Y coordinate :<br>=====                    | A point source is introduced by specifying the X and Y coordinates of the location of the point ("grid") in the model co-ordinate system. A spatially distributed source is introduced by specifying the spatial distribution as a dfs2 file with code '1' in the source area and '0' elsewhere. Source strengths are specified in the Species Dependent input part.   |
| RIVER INPUT PART<br>----- River dispersion<br>disp. coeff. : 0.0   | <b>These lines are ignored.</b> Solute transport in the river is now simulated in MIKE 11 and all input data is specified in the MIKE 11 AD GUI. .   |
| ----- Source locations<br>No of data elements : 2<br>river source location : 1<br>river source type : 1<br>river source Q-station : 1<br>river source location : 2<br>river source type : 1<br>river source Q-station : 6<br>=====   | For the River sources only type 1 is allowed which implies a mass flux with unit [mass/time] (base unit $\text{kg/s}$ ). The source location is given an identification code. The location is the Q-station number. The river source strength is given in the species dependent part of the input.   |
| PRECIPITATION/INFILTRATION INPUT PART<br>----- Source locations<br>No of data elements: 2<br>Location number : 1<br>Source type : 1<br>Area distribution : MAPS\precipitationSourceLocation.dfs2 1<br>Location number : 2<br>Source type : 1<br>Area distribution : MAPS\precipitationSourceLocation.dfs2 2<br>+++++ | <p>The most "natural" way of specifying a source in MIKE SHE AD is to give the precipitation a concentration and let the model itself calculate how this is distributed in the system. This option is also valid if you have only included the groundwater part in the simulation. The source will then be treated as an infiltration source instead.</p> <p>The precipitation is specified in MIKE SHE WM as a combination of the spatial distribution and the temporal variation. A "precipitation source" is specified in a similar way as the extend of each source is determined by its spatial distribution (a dfs2 file with the value '1' in grids included in the source and '0' elsewhere) and its concentration is specified in a time series data file (in the Species Dependent input part).</p> <p>Enter the location number (1, 2, etc.), the source type (the only valid type is 1 corresponding to a time varying concentration) and the spatial distribution as a dfs2 file.</p> |



*Table 13.4 Possible combinations of sources in the groundwater*

| Areal distribution                | Source Type in input | Source description                         |
|-----------------------------------|----------------------|--|
| point or line (over depth) source | 1                    | mass flux in a specific grid               |
|                                   | 3                    | fixed concentration in a specific grid     |
| spatially distributed source      | 2                    | time varying mass flux over a certain area |
|                                   | 4                    | fixed concentration over a certain area    |

*Table 13.5 Possible combinations of sources in the unsaturated zone*

| Areal distribution                | Source Type in input | Base Unit             | Source description                         |
|-----------------------------------|----------------------|-----------------------|--|
| point or line (over depth) source | 1                    | [kg/s]                | mass flux in a specific grid               |
| spatially distributed source      | 2                    | [g/m <sup>2</sup> /s] | time varying mass flux over a certain area |

*Table 13.6 Possible combinations of sources in the overland component*

| Areal Distribution           | Source Type in input | Source Description                         |
|------------------------------|----------------------|--|
| point source                 | 1                    | mass flux in a specific grid               |
| spatially distributed source | 2                    | time varying mass flux over a certain area |

### 13.3.2 Species *DEPENDENT* Data

As MIKE SHE is able to handle simultaneous transport of several species some of the parameters have to be specified for each of the species. These parameters include the initial concentrations, the source strengths for the sources specified in the species independent input part and some special parameters for groundwater and surface water transport.

• •

*Table 13.7 MIKE SHE AD .tsf file format and description continued*

| Line item  | Comment   |
|--|---|
| <pre> +++++*+++++*+++++*+++++*+++++*+++++*+++++* SPECIES DEPENDENT DATA +++++*+++++*+++++*+++++*+++++*+++++*+++++* ----- Species Species name      : Partikel </pre>   | <p>The first important item is to identify your species with their names. A species name is the only identification and must be given. Any name can be used for the species e.g. Chloride or Bromide but also names such as Species A or solute are possible. All the species dependent input has to be repeated for each species.</p>  |
| <pre> ===== SZ INPUT PART ----- Initial Conditions No of data elements: 3 Lower layer      : 1 Concentration    : MAPS\SZConcentration.dfs2 1 Lower layer      : 2 Concentration    : MAPS\SZConcentration.dfs2 2 Lower layer      : 8 Concentration    : 0 </pre> | <p>The initial condition is the initial concentration of the current species and can be either a constant value or fully distributed. The input consists of one or more data elements, where each data element consists of a lower layer indicating the layer down to which the input is valid and a concentration. The lower layer of the last data element must be equal to or larger than the lowest layer number in the model. The layer intervals in the data elements may not overlap.</p> <p>If a constant value is specified the unit of this value is the Base Unit for Concentration <math>\mu\text{g}/\text{m}^3</math>.</p> <p>If a dsf2 file is specified, then the units of concentration are part of the EUM system and stored in the dfs2 file. However, there is a minimum concentration in MIKE SHE, because the internal calculations are done in <math>\text{g}/\text{m}^3</math>. If the initial concentration is below this minimum, then it is set to zero.</p> <p><b>Particle tracking</b></p> <p>The initial concentration is also used to specify the initial concentration of particles for the PT module. In this case, the meaning of the concentration value depends on the value of INITSPEC. However, the initial concentration is read before INITSPEC, so the initial interpretation is as an initial concentration value.</p> <p>At run time, if INITSPEC is -1, then the initial concentration is converted to a number of particles for the PT module. Thus, when using INITSPEC = -1, the initial concentration must be greater than the minimum, or else the initial number of particles will be zero.</p> |



Table 13.7 MIKE SHE AD .tsf file format and description continued

| Line item   | Comment   |
|---|---|
| <p>----- Source strength</p> <p>No of data elements: 2</p> <p>Location number : 1</p> <p>Time serie : 100.0</p> <p>Record number :</p> <p>Location number : 2</p> <p>Time serie : TIME\SZSourceStrength.dfs0</p> <p>Record number : 2</p> | <p>In the species dependent section input is given for the strength of the SZ sources for which the location was specified in the species independent section. The unit for the source strength depends on the source type. The input can either be a constant (time invariant) input in which case the unit depends on the Base unit for the data type or a time series in a dfs0 time series file in which case the item has to have the correct eum data type.</p> <p>The input consists of a number of data elements, one for each source location. The data element contains a Location Number for identification purposes and then two lines with the value or time series file name and in case a time series file is used the item number.</p>  |
| <p>----- Dual Porosity Mass Transfer Coeff</p> <p>No of data elements : 1</p> <p>Lower layer :99</p> <p>Concentration :MAPS\SZTransferCoefficient.dfs2 1</p>  | <p>Solute transport in dual porosity media e.g. chalk aquifers takes place both in the fractures and in the aquifer matrix. The exchange of mass between the fractures and the matrix is described by a diffusion process and the mass transfer coefficient controls rate of solute exchange between the two phases. As this coefficient is increased, solute diffusion takes place at a faster rate which causes lower peaks but a slower attenuation of the peak in a concentration break through curve. It is an empirical constant and cannot be compared directly with the diffusion coefficient for the species.</p> <p>Since the mass transfer coefficient is an empirical constant and varies both with the characteristics of the species and of the media it is difficult to recommend its range. The Eum data type for this input is set to eumIFstOrder-RateWQ for which the base unit is day<sup>-1</sup>. In applications it has been calibrated to about 1 x 10<sup>-12</sup> sec.<sup>-1</sup>. The input is like for the initial concentrations and can be fully distributed. The input is only needed if the double porosity option is included which is set in the general section of the input.</p> |
| <p>=====</p> <p>UZ INPUT PART</p> <p>----- Initial conditions</p> <p>No of data elements: 1</p> <p>Lower depth : 999.</p> <p>Concentration : 0</p>  | <p>The initial concentration can be fully distributed over the unsaturated zone in the catchment and is specified using a number of data elements where each of these consists of a lower depth and a concentration either given as a uniform value or as a distributed value in a dfs2 file: The value specified in a data element applies from the soil surface - in case it's the first input - or the lower depth of the previous data element to the lower depth for the data element. The depths are specified in meter below ground (m). The following input could be used to give an initial concentration of 0.0 to all UZ compartments. The unit when using a constant value is the Base Unit for concentration µg/m<sup>3</sup>.</p>   |





Table 13.7 MIKE SHE AD .tsf file format and description continued

| Line item  | Comment   |
|--|---|
| <p>----- Source strength</p> <p>No of data elements: 2</p> <p>Location number : 1</p> <p>Time series : TIME\UZSourceStrength.dfs0</p> <p>Record number : 1</p> <p>Location number : 2</p> <p>Time series : 200</p> <p>Record number :</p>                          | <p>The units for the source strength depend on the source type. Each of the sources identified by their location number in the species independent section must be given a strength.</p> <p>The source strength can either be constant during the entire simulation period in which case the unit is the Base unit or can be a dfs0 time series in which case the item has to have the correct EUM data type and the user is free to set the unit to one of the available units for the eum data type.</p> <p>The input consists of a number of data elements, one for each UZ source location. The data element has a Location Number for identification purposes and then two lines with the value or time series file name and in case a time series file is used the item (record) number.</p>                            |
| <p>=====</p> <p>OVERLAND INPUT PART</p> <p>----- Initial Conc. and Solubility</p> <p>Initial Conditions : MAPS\OLConcentration.dfs2 2</p> <p>Solubility : 100</p>  | <p>The initial concentration input for the overland is given as surface concentration [M/L<sup>2</sup>]. In this way it is easier to control the mass of solutes introduced because you do not have to consider surface water depth. It can be given as single value in which case the unit is the base unit for the data type eumIMassPerUnitArea (Mass Per unit Area) g/m<sup>2</sup> or as a distributed value in a dfs2 file. As evaporation can cause the overland concentration to increase solubility needs to be specified to avoid unrealistic high values. The species precipitates if the concentration exceeds the solubility and the solid dissolves again if the concentration decreases below the solubility. The solubility is a uniform value for the entire catchment and has the unit g/m<sup>2</sup>.</p> |
| <p>----- Source strength</p> <p>No of data elements: 2</p> <p>Location number : 1</p> <p>Time series : TIME\OLSourceStrength.dfs0</p> <p>Record number : 3</p> <p>Location number : 2</p> <p>Time series : TIME\OLSourceStrength.dfs0</p> <p>Record number : 4</p> | <p>The units for the source strengths depend on the source type. Each of the sources identified by their location number in the species independent section must be given a strength.</p> <p>The source strength can either be constant during the entire simulation period in which case the unit is the Base unit or can be a dfs0 time series in which case the item has to have the correct EUM data type and the user is free to set the unit to one of the available units for the eum data type.</p> <p>The input consists of a number of data elements, one for each overland source location. The data element has a Location Number for identification purposes and then two lines with the value or time series file name and in case a time series file is used the item (record) number.</p>                     |



Table 13.7 MIKE SHE AD .tsf file format and description continued

| Line item   | Comment  |
|---|--|
| <p>=====</p> <p>RIVER INPUT PART</p> <p>----- Initial Conditions</p> <p>Initial conc. : 0.0</p>   | <p><b>Note:</b> If MIKE 11 AD is used, these lines are ignored.</p> <p>The input for the river system applies only to the MIKE SHE AD river module. The initial concentration of the water in the river system can only be given as a uniform value i.e. the value is applied throughout the entire river network. The unit is the base unit for concentration and is <math>\mu\text{g}/\text{m}^3</math>.</p>   |
| <p>----- Source Strength</p> <p>No of data elements: 2</p> <p>Location number : 1</p> <p>Time series : TIME\SourceStrength.dfs0</p> <p>Record number : 5</p> <p>Location number : 2</p> <p>Time series : 1.e-6</p> <p>Record number :</p>   | <p>With respect to the source strenghts these can only be given as mass fluxes. The source strength can be constant during the entire simulation period in which case a single value is specified with the unit is kg/s time which is the base unit for a mass flux. If the source strength is time varying the name of the dfs0 is specified and the record number corresponding to the relevant time series. The item in the dfs0 file has to have the data type eumISolute-Flux (Solute flux). The input consists of a number of data elements, one for each overland source location. The data element has a Location Number for identification purposes and then two lines with the value or time series file name and in case a time series file is used the item (record) number.</p> |
| <p>=====</p> <p>PRECIPITATION/INFILTRATION INPUT PART</p> <p>----- Source strength</p> <p>No of data elements: 2</p> <p>Location number : 1</p> <p>Time series : TIME\PrecipitaionSourceStrength.dfs0</p> <p>Record number : 5</p> <p>Location number : 2</p> <p>Time series : 5.0</p> <p>Record number :</p> | <p>The last input item in the species dependent section is the precipitation sources strength. The source concentration can be given as a time invariant constant value or a time series. In case a constant value is used the unit is the base unit for concentration <math>\mu\text{g}/\text{m}^3</math>. In case a dfs0 file is specified the item in the file has the data type EumIConcentration.</p> <p>The input consists of a number of data elements, one for each overland source location. The data element has a Location Number for identification purposes and then two lines with the value or time series file name and in case a time series file is used the item (record) number.</p>   |
| <p>+*+*+*+*+*+*+*+*+*+*+*+*+*+*+*</p> <p>EXTRA TSF FILE : F</p>   | <p><b>EXTRA PARAMETER SECTION</b></p> <p>Input for newer options is given at the bottom of the tsf file.</p> <p>The sorption degradation module for MIKE SHE AD reads its input from a separate tsf file with extension xtsf. The option can be activated by adding the following line and setting the input to T:</p>   |



Table 13.7 MIKE SHE AD .tsf file format and description continued

| Line item  | Comment  |
|--|--|
| MIKE11 AD : T<br>MIKE11 FILE NAME : MIKE11\Karup_AD_SR.sim11<br>MIKE11 time step : 30<br>DISABLE VALIDATION OF HD RES FILE : T | <p>For the river module MIKE SHE AD can also be linked to the MIKE 11 AD. To use the option the user has to run the water movement with the MIKE 11. The input for the AD coupling consists of three lines. The MIKE 11 time step has the unit seconds in the tsf file. During initialisation MIKE SHE AD checks if the MIKE 11 result for the water movement is consistent with the MIKE SHE water movement result by checking a timestamp that is stored with water movement results. This check can be switched off by setting the validation to false (F).</p> <p>The last line should always be true and the only input given should be the name of the sim11 file.</p> |
| DISABLE TIMER OUTPUT : T   | In the print file the MIKE SHE AD stores information on run time statistics. This output can be disabled.  |

Table 13.8 Combinations of source types, EUM data types and units in the groundwater transport component

| SZ Source Type | EUM data type (user description)                               | Base unit             | Source description                         |
|----------------|--|-----------------------|--|
| 1              | eumISoluteFlux (Solute flux)                                   | [kg/s]                | mass flux in a specific grid               |
| 3              | eumIConcentration (Concentration)                              | [µg/m <sup>3</sup> ]  | fixed concentration in a specific grid     |
| 2              | eumISpecificSoluteFlux-PerArea (Specific Solute Flux Per Area) | [g/m <sup>2</sup> /s] | time varying mass flux over a certain area |
| 4              | eumIConcentration (Concentration)                              | [µg/m <sup>3</sup> ]  | fixed concentration over a certain area    |

Table 13.9 Combinations of source types, EUM data types and units in the unsaturated zone transport component

| UZ Source Type | EUM data type (user description)                               | Base unit             | Source description                         |
|----------------|--|-----------------------|--|
| 1              | EUMISoluteFlux (Solute flux)                                   | [kg/s]                | mass flux in a specific grid               |
| 2              | EumISpecificSolute-FluxPerArea (Specific Solute Flux Per Area) | [g/m <sup>2</sup> /s] | time varying mass flux over a certain area |



Table 13.10 Combinations of source types, EUM data types and units in the overland transport component

| UZ Source Type | EUM data type (user description)                              | Base unit             | Source description                         |
|----------------|---|-----------------------|--|
| 1              | EUMISoluteFlux (Solute flux)                                  | [kg/s]                | mass flux in a specific grid               |
| 2              | EumISpecificSoluteFluxPerArea (Specific Solute Flux Per Area) | [g/m <sup>2</sup> /s] | time varying mass flux over a certain area |

## 13.4 Working with the .XTSF File

The extra input data file for a simulation with the AD module is called the extra transport set-up file - xtsf. This is also a text file, which contains all relevant information for the simulation reactions included in the sorption-degradation module of MIKE SHE AD. The file must be present in the catchment working directory with the name <set-up.xtsf>.

The xtsf file is divided into three parts:

- simulation parameters,
- species dependent parameters, and
- process dependent parameters.

The file contains one or more of the following parameters depending on the type of processes included (Base Units are given in parentheses, which must be used if constant values are input):

### Sorption

$\rho_b$ : Bulk density (kg/m<sup>3</sup>)

Linear isotherm:

$K_d$ : Linear sorption coefficient (m<sup>3</sup>/g)

Freundlich isotherm:

$K_f$ : Freundlich sorption coefficient (m<sup>3</sup>/g)

$N$ : Exponent, which may vary between 0 and 1.

Langmuir isotherm:

$\alpha$ : Constant related to the binding energy.

$\beta$ : Maximum amount of sorbed solute.



Kinetic sorption: The kinetic approach may be used together with each of the isotherms. For example, if the linear sorption isotherm is used half of the  $K_d$  value is assumed to in equilibrium, the other half is assumed to be kinetic.

$K1$ : Rate constant ( $s^{-1}$ ). If hysteresis is included two rate constant must be given..

Table 13.11 Input parameters for the different isotherms

| Xtsf-file              | Linear   | Freundlich   | Langmuir   |
|------------------------|--|--|--|
| <b>K1</b>              | $K_d$<br>EUM Data type: First Order<br>Grazing Rate Dependence<br>Base Unit: $m^3/g$ | $K_f$<br>EUM Data type: First Order<br>Grazing Rate Dependence<br>Base Unit: $m^3/g$ | $\alpha$   |
| <b>K2</b>              | $\theta$   | $N$<br>EUM Data type:<br>Dimensionless exponent<br>Base Unit: -                      | $\beta$  |
| <b>K3 (if kinetic)</b> | Sorption rate<br>EUM Data type:<br>1st order rate WQ model<br>Base Unit: 1/day       | Sorption rate<br>EUM Data type:<br>1st order rate WQ model<br>Base Unit: 1/day       | Sorption rate<br>EUM Data type:<br>1st order rate WQ model<br>Base Unit: 1/day   |
| <b>K4</b>              | Desorption rate<br>EUM Data type:<br>1st order rate WQ model<br>Base Unit: 1/day     | Desorption rate<br>EUM Data type:<br>1st order rate WQ model<br>Base Unit: 1/day     | Desorption rate<br>EUM Data type:<br>1st order rate WQ model<br>Base Unit: 1/day |

### Sorption in dual porosity systems

$F_b$ : Bias factor accounting for the distribution of sorption sites between matrix and macro pores may vary between -1 and +1.

### Decay

$\lambda$ : Half life time (s).

$B$ : Empirical exponent accounting for the effect of soil moisture conditions on the degradation rate (e.g. 0.7).

$T_{ref}$ : Reference temperature ( $^{\circ}C$ ) at which the given half life time was measured.

$\alpha$ : Constant accounting for the effect of soil temperature on the degradation rate (e.g. 0.08).

### Plant uptake

$f_c$ : Concentration factor accounting for extent of plant uptake. May vary between 0 and 1.



### 13.4.1 Simulation Parameters

The simulation parameter input part of the .xtsf file should always be included. This section contains information about:

- Number of processes;
- Logical options for including sorption, decay, soil temperature and plant uptake;
- Air temperature;
- Bulk density distribution;
- Initial soil temperature distribution

Table 13.12 MIKE SHE AD .xtsf file format and description

| Line item  | Comment  |
|--|--|
| <pre> FILETYPE DATA TYPE Verno: 2002 1 524 ===== EXTRA SETUP DATA for Transport Sim ===== PROCESSES IN CALCULATION No. of processes      : 3 ===== ----- Extensions Sorption              : T Decay                 : T Soil temperature      : T Plant uptake          : T </pre> | <p>The xtsf file starts with a header giving the information on the type and version of the input file. The current version should be 524.</p> <p>The next lines are for information only.</p> <p>The next lines specifies the processes included in the simulation, how many processes and which ones:</p>  |
| <pre> ===== FR INPUT PART ----- Air temperature Air temp grid codes: 1 Air temp tim.ser    : TIME/tempture.dfs0 ===== </pre>   | <p>The next lines specify the temperature distribution in space and time, which is relevant for the plant uptake simulation. The temperature is specified as a combination of the spatial distribution and the temporal variation, i.e. a dfs2 file (or constant value) with grid codes determining the distribution of the temperature stations and the temperature specified in a time series data files. Remember to specify the record number if the distribution is given as a dfs2 file.</p> |



Table 13.12 MIKE SHE AD .xtsf file format and description

| Line item   | Comment   |
|---|---|
| <pre> SZ INPUT PART ----- Bulk density No of data elements: 1 Lower layer      : 999 Bulk density     : 1600.0 ----- Initial soil temperature No of data elements: 1 Lower layer      : 999 Init. soil temp. : 6.0 ===== </pre>   | <p>The next lines specify the bulk density and initial soil temperature for the saturated groundwater zone (SZ). The input consists of one or more data elements. Each data element consists of a lower layer input indicating the layer down to which the input is valid and a concentration input. The data elements should cover all layers in the model i.e. the last data element's Lower layer input should be equal to or larger than the lowest layer number. The layer intervals in the data elements may also not overlap.</p> <p><b>Bulk Density</b><br/>EUM Data type: Density<br/>Base Unit: kg/m<sup>3</sup></p> <p><b>Soil Temperature</b><br/>EUM Data type: Temperature<br/>Base Unit: Celcius</p>   |
| <pre> UZ INPUT PART ----- Bulk density No of data elements: 1 depth              : 999. Bulk density       : MAPS\bulkden- sity.dfs2 1 ----- Initial soil temperature No of data elements: 1 Depth             : 999.0 Init. soil temp.   : MAPS\inTemp.dfs2 1 +++++ </pre> | <p>The next lines specify the bulk density and initial soil temperature for the unsaturated zone (UZ). These parameters are specified using a number of data elements where each of these consists of a lower depth and a bulk density/temperature either given as a uniform value or as a distributed value in a dfs2 file: The value specified in a data element applies from the soil surface - in case it's the first input - or the lower depth of the previous data element to the lower depth for the data element.</p> <p><b>Bulk Density</b><br/>EUM Data type: Density<br/>Base Unit: kg/m<sup>3</sup></p> <p><b>Soil Temperature</b><br/>EUM Data type: Temperature<br/>Base Unit: Celcius</p> <p>The depths are specified in meters below ground (m).</p> |



Table 13.12 MIKE SHE AD .xtsf file format and description

| Line item  | Comment  |
|--|--|
| SPECIES DEPENDENT DATA<br>+++++<br>----- Species<br>Species name : Solute<br>===== FR INPUT PART<br>--- Plant uptake transpiration factor<br>transp.factor (0-1): 1.0<br>===== UZ INPUT PART<br>-- Matrix-Macropore Mass Transfer Coeff<br>No of data elements: 0<br>+++++<br>----- Species<br>Species name : Sorbed-Solute<br>===== FR INPUT PART<br>-- Plant uptake transpiration factor<br>transp.factor (0-1): 0.0<br>===== UZ INPUT PART<br>-- Matrix-Macropore Mass Transfer Coeff<br>No of data elements: 0<br>+++++<br>----- Species<br>Species name : degradable<br>===== FR INPUT PART<br>---Plant uptake transpiration factor<br>transp.factor (0-1): 0.5<br>===== UZ INPUT PART<br>- Matrix-Macropore Mass Transfer Coeff<br>No of data elements: 0<br>+++++ | <p>For each species defined in the .tsf file this section must contain the following parameters:</p> <p>Plant uptake transpiration factor - which determines which fraction of the concentration of the solute the plant takes up.</p> <p>Matrix-macro pore mass transfer coefficient - which is given as for dual porosity diffusion in the groundwater system.</p> |





Table 13.12 MIKE SHE AD .xtsf file format and description

| Line item   | Comment  |
|---|--|
| <pre> PROCESS DEPENDENT DATA +++++ --- Process identification Process name      : sorption Proc. type 1/2/3/4 : 3 ===== </pre>  | <p>For each of the processes defined a set of process dependent parameters must be defined for UZ and SZ:</p> <p>The input section for a sorption process must contain a selection of sorption isotherm and related parameters, which may be distributed in depth. For each isotherm the two input lines K1 and K2 should always be filled out whereas K3 and K4 are only necessary when sorption kinetics is included. See Table 13.11.</p> <p>The input section for a decay process must contain information on dependency of soil temperature and soil moisture content together with half-life-times distributed in depth. If macro pore transport is simulated, different decay parameters can be chosen for each domain.</p> <p>The next lines in the .xtsf file specifies the process dependent parameters. For each process the parameters are given as general governing parameters, groundwater (SZ) related parameters and unsaturated zone (UZ) related parameters.</p> <p>The first process begins with the name of the process and the process type (1/2/3/4) i.e. 3 is sorption and 4 is degradation.</p> |
| <pre> GENERAL PROCESS PARAMETERS ---- Species definitions Dissolved species  : Solute Sorbed species     : Sorbed-Solute ---- Type of sorption description EQ/EQ+KIN 1/2      : 1 --- Type of Equilibrium-sorption Lin/Freu/Lang 1/2/3: 1 ----- Type of Kinetic-sorption No Hyst./Hyst. 1/2 : 1 - Fracture sorption Bias factor (only for DP) Frac-sorp bias fac : 0. ---- EQ-sorption fraction EQ-sorp frac [0-1] : 0.5 ===== </pre> | <p>Then the general process parameters are given defining which species are included in the process and what type of sorption description applies. In the example we have chosen equilibrium sorption, linear without hysteresis.</p>  |



Table 13.12 MIKE SHE AD .xtsf file format and description

| Line item  | Comment  |
|--|--|
| SZ-RELATED PROCESS PARAMETERS<br>----- Sorption coefficients<br>No of data elements: 1<br>Lower layer : 999<br>K1 (EQ) : MAPS\Kd.dfs2 1<br>K2 (EQ) : 0.0<br>K3 (KIN) : 0.0<br>K4 (KIN) (hyst.) : 0.0<br>=====  | <p>The next lines specify the sorption coefficients (K1 to K4) valid in the groundwater for the relevant sorption description as shown in Table 13.11. These parameters are specified in a similar manner as e.g. initial concentrations. The input consists of one or more data elements. Each data element consists of a lower layer input indicating the layer down to which the input is valid and the four parameters. The data elements should cover all layers in the model i.e. the last data element's Lower layer input should be equal to or larger than the lowest layer number. The layer intervals in the data elements may also not overlap. If a constant value is specified the Base Unit for the sorption parameter applies. If a dfs2 file is used, you specify the unit in the file.</p> |
| UZ-RELATED PROCESS PARAMETERS<br>----- Sorption coefficients<br>No of data elements: 1<br>Depth : 99.0<br>K1 (EQ) : 1e-6<br>K2 (EQ) : 0.0<br>K3 (KIN) : 0.0<br>K4 (KIN) (hyst.) : 0.0<br>+++++   | <p>The next lines specify the sorption coefficients (K1, .. , K4) valid in the unsaturated zone for the relevant sorption description as shown in Table 13.11. These parameters are specified using a number of data elements where each of these consists of a lower depth and the sorption constant either given as a uniform value or as a distributed value in a dfs2 file. The value specified in a data element applies from the soil surface - in case it's the first input - or the lower depth of the previous data element to the lower depth for the data element. The depths are specified in metres below ground (m).</p>   |
| +++++<br>+++++<br>----- Process identification<br>Process name : Decay<br>Proc. type 1/2/3/4 : 4<br>=====  | <p>The second process begins also with the process identification is specified with its name and the process type (1/2/3/4) - in this case degradation (4). The process in this example is decay, which depends on temperature and water content in the unsaturated zone.</p>  |
| GENERAL PROCESS PARAMETERS<br>----- Species definitions<br>Current species : degradable<br>----- Temperature dependent decay<br>Temp-dep.decay T/F : T<br>Ref. temp (deg.C) : 10.0<br>Temp-decay exponent: 0.079<br>----- Water content dependent decay<br>WC-dep. decay T/F : T<br>WC-decay exponent : 0.7<br>===== | <p>Then follows the lines about the general process parameters, defining which species are included in the process and what type of sorption description applies. The reference temperature <math>T_{ref}</math> for the temperature dependent decay calculation is given in degrees centigrade and the exponent <math>\alpha</math> is dimensionless. The exponent for water content dependent decay, <math>B</math>, is also dimensionless.</p>  |



Table 13.12 MIKE SHE AD .xtsf file format and description

| Line item  | Comment  |
|--|--|
| SZ-RELATED PROCESS PARAMETERS<br>----- Half-life time<br>No of data elements: 1<br>Lower layer : 999<br>Sngl-por:T1/2(sec.): 864000.0<br>DP:T1/2,frac (sec.): 864000.0<br>DP:T1/2,matr (sec.): 864000.0<br>=====   | <p>The next lines specify the decay process in the groundwater in terms of half-life times. Note that it is necessary to specify both for porous media transport and fractured transport in both fractures and matrix. These parameters are specified in a similar manner as e.g. initial concentrations. The input consists of one or more data elements. Each data element consists of a lower layer input indicating the layer down to which the input is valid and the four parameters. The data elements should cover all layers in the model i.e. the last data element's Lower layer input should be equal to or larger than the lowest layer number. The layer intervals in the data elements may also not overlap. If a constant value is specified the Base Unit for the sorption parameter applies. If a dfs2 file is used, you specifies the unit in the file.</p> <p><b>Half life time</b><br/>           EUM Data type: Time scale<br/>           Base Unit: Seconds</p> |
| UZ-RELATED PROCESS PARAMETERS<br>----- Half-life time<br>No of data elements: 2<br>Depth : 0.55<br>Sngl-por:T1/2(sec.): MAPS\tc.dfs2 1<br>DP:T1/2,frac (sec.):<br>DP:T1/2,matr (sec.):<br>Depth : 999.0<br>Sngl-por:T1/2(sec.): 0.0<br>DP:T1/2,frac (sec.): 0.0<br>DP:T1/2,matr (sec.): 0.0<br>===== | <p>In the next lines specifies the decay parameters valid in the unsaturated zone. These parameters are specified using a number of data elements where each of these consists of a lower depth and the sorption constant either given as a uniform value or as a distributed value in a dfs2 file. The value specified in a data element applies from the soil surface - in case it's the first input - or the lower depth of the previous data element to the lower depth for the data element. The depths are specified in meter below ground (m).</p> <p><b>Half life time</b><br/>           EUM Data type: Time scale<br/>           Base Unit: Seconds</p>  |
| OL-RELATED PROCESS PARAMETERS<br>----- Half-life time<br>T1/2(sec.) : 0.<br>++++++   | <p>Decay may also apply for the solutes in overland water and the next lines specify the half-life time for the overland component.</p>  |

## 13.5 Example input files

The following files can be found in the Examples sub-directory in your installation directory.

### 13.5.1 Transport Setup File (\*.tsf)

```

FILETYPE DATA TYPE Verno:    2001        1        524
=====
SETUP DATA for Transport Simulation

```



```
=====
THIS FILE OK (T/F) : T
=====
RESULTS OF WATERMOVEMENT CALCULATION
file name           : Aarhus_n02_sce1\Aarhus_n02_sce1.frf
=====
SPECIES IN CALCULATION
No. of species      : 1
=====
SIMULATION CONTROL PARAMETERS
----- Simulation Identification
Simulation title    : Aarhus Nord02 PT
Simulation descr    : Aktuel
----- Simulation period
Cycle Option 1/2/3 : 2
Start year         : 1981
Start month        : 12
Start day          : 26
Start hour         : 0
Start minute       : 0
End year           : 2997
End month          : 12
End day            : 31
End hour           : 0
End minute         : 0
Cycle restart year : 1981
Cycle restart month: 12
Cycle restart day  : 26
Cycle restart hour : 0
Cycle restart min  : 0
Cycle end year     : 2001
Cycle end month    : 12
Cycle end day      : 11
Cycle end hour     : 0
Cycle end minute   : 0
Stat. Flow year    :
Stat. Flow month   :
Stat. Flow day     :
Stat. Flow hour    :
Stat. Flow minute  :
----- Execution Logicals
Execute SZ      T/F : T
Execute All.   T/F : F
----- Extensions
SZ Double Por. : F
----- Inclusions of sources
SZ              T/F : F
UZ              T/F : F
Overland       T/F : F
River          T/F : F
Prec./Inf.     T/F : F
----- Storing of Results
Storing SZ      T/F: T
```



```

Storing UZ          T/F: F
Storing OverlandT/F: F
Storing River      T/F: F
Storing MBL r      T/F: F
St.Frq  SZ          : 175200.000000
St.Frq  UZ          : 720.000000
St.Frq  Overland    : 240.000000
St.Frq  River       : 24.000000
St.Frq  MassBal.    : 720.000000
Print frq MBL       : 43800.000000
----- Time Step Control
Max SimTimeStep SZ : 1000000000.000000
Max SimTimeStep UZ : 1000000000.000000
Max SimTimeStep Ovl: 1000000000.000000
Max SimTimeStep Riv: 1000000000.000000
Max Advec.Cour. SZ : 0.800000
Max Advec.Cour. UZ : 0.800000
Max Advec.Cour.Ovl : 0.800000
Max Advec.Cour.Riv : 0.800000
Max Disp.Cour. SZ  : 0.500000
Max Disp.Cour. UZ  : 0.500000
Max Disp.Cour. Ovl : 0.500000
Max Disp.Cour.Riv  : 0.500000
Max Transport SZ   : 0.950000
Max Transport UZ   : 0.950000
Max Transport Overl: 0.950000
Max Transport River: 0.950000
=====
SZ INPUT PART
----- Effective porosity
No of data elements: 1
Lower layer        : 99
Porosity           : .15
----- Type of dispersion description
Aniso. opt. 1/2/3  : 1
----- Dispersivities
No of data elements: 1
Lower Layer       : 99
Disp. alfa LHH    : 2.
Disp. alfa THH    : .2
Disp. alfa TVH    :
Disp. alfa LVV    :
Disp. alfa THV    :
----- Matrix porosity
No of data elements: 0
----- Source locations
No of data elements: 0
=====
UZ INPUT PART
----- Dispersivities
No of data elements: 0
----- Source locations
No of data elements: 0

```



```
=====
OVERLAND INPUT PART
----- Dispersion
X-disp. coeff.      :
Y-disp. coeff.      :
----- Source locations
No of data elements: 0
=====
RIVER INPUT PART
----- River dispersion
disp. coeff.        :
----- Source locations
No of data elements: 0
PRECIPITATION/INFILTRATION INPUT PART
----- Source locations
No of data elements: 0
+++++
SPECIES DEPENDENT DATA
+++++
----- Species
Species name         : Partikel
=====
SZ INPUT PART
----- Initial Conditions
No of data elements: 6
Lower layer          : 1
Concentration         : 0
Lower layer          : 2
Concentration         : PT\partikler100.dfs2 1
Lower layer          : 3
Concentration         : PT\partikler10.dfs2 1
Lower layer          : 4
Concentration         : PT\partikler10.dfs2 1
Lower layer          : 5
Concentration         : 0
Lower layer          : 6
Concentration         : PT\partikler10.dfs2 1
----- Source strength
No of data elements: 0
----- Dual Porosity Mass Transfer Coeff
No of data elements: 0
=====
Z INPUT PART
----- Initial conditions
No of data elements: 0
----- Source strength
No of data elements: 0
=====
OVERLAND INPUT PART
----- Initial Conc. and Solubility
Initial Conditions   :
Solubility           :
----- Source strength
```



```

No of data elements: 0
=====
RIVER INPUT PART
----- Initial Conditions
Initial conc.      :
----- Source Strength
No of data elements: 0
=====
PRECIPITATION/INFILTRATION INPUT PART
----- Source strength
No of data elements: 0
+*****+
PARTICLE TRACKING : T
PARTICLE MASS     : 100.0
INITSPEC          : -1
VERTICAL CORRECTION : 1.0
CONSTANT CELL PARTICLES: F
ONLY SATURATED : T
DISTRIBUTION TYPE : 1
RELATIVE INITIAL LEVEL : 0.999
REGZONEFILE       : none
DFS2 INPUT        : F
WELLREGISTRATION  : T
LPTBIN            : F
DISABLE TIMER OUTPUT : T

```

### 13.5.2 Extra transport setup file (.xtsf)

```

FILETYPE DATA TYPE Verno: 2002      1      600
=====
EXTRA SETUP DATA for Transport Simulation
=====
PROCESSES IN CALCULATION
No. of processes      : 2
=====
----- Extensions
Sorption              : T
Decay                 : T
Soil temperature      : T
Plant uptake          : T
=====
FR INPUT PART
----- Air temperature
Air temp grid codes: 1
Air temp tim.ser     : TIME/temperature.dfs0
=====
SZ INPUT PART
----- Bulk density
No of data elements: 1
Lower layer         : 999
Bulk density        : 1600.0
----- Initial soil temperature
No of data elements: 1

```



```
Lower layer      : 999
Init. soil temp. : 6.0
=====
UZ INPUT PART
----- Bulk density
No of data elements: 1
depth              : 999.
Bulk density       : MAPS\bulkdensity.dfs2 1
----- Initial soil temperature
No of data elements: 1
Depth              : 999.0
Init. soil temp.   : MAPS\initialTemp.dfs2 1
+++++
SPECIES DEPENDENT DATA
+++++
----- Species
Species name       : Solute
=====
FR INPUT PART
----- Plant uptake transpiration factor
transp.factor (0-1): 1.0
=====
UZ INPUT PART
----- Matrix-Macropore Mass Transfer Coeff
No of data elements: 0
+++++
----- Species
Species name       : Sorbed-Solute
=====
FR INPUT PART
----- Plant uptake transpiration factor
transp.factor (0-1): 0.0
=====
UZ INPUT PART
----- Matrix-Macropore Mass Transfer Coeff
No of data elements: 0
+++++
----- Species
Species name       : degradable
=====
FR INPUT PART
----- Plant uptake transpiration factor
transp.factor (0-1): 0.5
=====
UZ INPUT PART
----- Matrix-Macropore Mass Transfer Coeff
No of data elements: 0
+++++
PROCESS DEPENDENT DATA
+++++
----- Process identification
Process name       : sorption
Proc. type 1/2/3/4 : 3
```





```

=====
GENERAL PROCESS PARAMETERS
----- Species definitions
Dissolved species   : Solute
Sorbed species      : Sorbed-Solute
----- Type of sorption description
EQ/EQ+KIN 1/2       : 1
----- Type of Equilibrium-sorption
Lin/Freu/Lang 1/2/3 : 1
----- Type of Kinetic-sorption
No Hyst./Hyst. 1/2  : 1
----- Fracture sorption Bias factor (only for
DP)
Frac-sorp bias fac  : 0.
----- EQ-sorption fraction
EQ-sorp frac [0-1]  : 0.5
=====
SZ-RELATED PROCESS PARAMETERS
----- Sorption coefficients
No of data elements: 1
Lower layer         : 999
K1 (EQ)             : MAPS\Kd.dfs2 1
K2 (EQ)             : 0.0
K3 (KIN)            : 0.0
K4 (KIN) (hyst.)    : 0.0
=====
UZ-RELATED PROCESS PARAMETERS
----- Sorption coefficients
No of data elements: 1
Depth              : 99.0
K1 (EQ)            : 1e-6
K2 (EQ)            : 0.0
K3 (KIN)           : 0.0
K4 (KIN) (hyst.)   : 0.0
+++++*****
----- Process identification
Process name        : Decay
Proc. type 1/2/3/4  : 4
=====
GENERAL PROCESS PARAMETERS
----- Species definitions
Current species     : degradable
----- Temperature dependent decay
Temp-dep.decay T/F  : T
Ref. temp (deg.C)   : 10.0
Temp-decay exponent: 0.079
----- Water content dependent decay
WC-dep. decay T/F   : T
WC-decay exponent   : 0.7
=====
SZ-RELATED PROCESS PARAMETERS
----- Half-life time
No of data elements: 1

```



```
Lower layer          : 999
Sngl-por:T1/2(sec.): 864000.0
DP:T1/2,frac (sec.): 864000.0
DP:T1/2,matr (sec.): 864000.0
=====
UZ-RELATED PROCESS PARAMETERS
----- Half-life time
No of data elements: 2
Depth              : 0.55
Sngl-por:T1/2(sec.): MAPS\tc.dfs2 1
DP:T1/2,frac (sec.):
DP:T1/2,matr (sec.):
Depth              : 999.0
Sngl-por:T1/2(sec.): 0.0
DP:T1/2,frac (sec.): 0.0
DP:T1/2,matr (sec.): 0.0
=====
OL-RELATED PROCESS PARAMETERS
----- Half-life time
T1/2(sec.)         : 0.
+++++
```



## **14 WORKING WITH PARTICLE TRACKING**

MIKE SHE particle tracking (PT) is in fact a part of MIKE SHE AD. PT allows the user to calculate the flow path of a number of hypothetical particles, which are moved in the three-dimensional, saturated groundwater zone (SZ). The particles are displaced individually in a number of time steps. The movement of each particle is composed of a deterministic part, in which the particle is moved according to the local ground water velocity calculated by the MIKE SHE water movement module, and a stochastic part where the particle is moved randomly based on the local dispersion coefficients.

Particle tracking is only calculated for the saturated zone (SZ) and particles that leave SZ are not traced any further. Initially, the user assigns a number of particles to the compartments of the model grid (the computational cells). Input of particles during the calculation can occur from sources in the precipitation or SZ or from boundary or internal constant concentration cells. Particles leave SZ when they arrive at a boundary or an internal constant concentration cell or when they go to a sink. Possible sinks in the Particle Tracking are wells, rivers, drains, and exchange with the unsaturated zone (UZ).

All particles are assigned a mass, which means that a number of particles within a specific volume correspond to a solute concentration. The Particle Tracking module can therefore be used for solute transport simulations and is in some cases superior to the conventional numerical solution of the advection-dispersion equation since numerical dispersion is negligible. However, it is expected that the module will be used mostly for delineation of abstraction well capture zones and upstream zones and for determination of groundwater age and solute transport times.

The PT module uses the concept of 'registration' cells. This is intended for recording particle data when particles enter certain model compartments. Registration cells can be used to delineate capture zones or to observe particles passing through some region of interest.

### **14.1 Requirements in MIKE SHE WM**

Prior to running a PT simulation, the MIKE SHE WM (Water Movement) simulation must be run. This chapter describes what needs to be specified in the WM simulation to run the PT simulation afterwards.



### 14.1.1 Flow Storing Requirements

Particle transport calculations in MIKE SHE PT are based on the ground-water flows from a MIKE SHE WM simulation. In principle, only ground-water fluxes are needed but to ensure that all the needed WM result data types are stored the user has to specify that results should be stored for an AD run in the WM input, i.e. tick the appropriate box under “Storing of Results” in the MIKE SHE WM GUI. The user can choose between “SZ only” and “All hydraulic components”, however, for PT-simulations “SZ only” will be sufficient, since particle tracking is only calculated for the saturated zone

The simulated particle distribution is stored with a desired frequency in the MIKE SHE WM GUI under “Storing of results” => “Grid series output”. It is important that the SZ and SZ-flow use the same storing frequency in order to run the following PT simulation. The WM result files to be used in the PT-simulations will be located in a folder with the same name as the \*.SHE file.

### 14.1.2 Specification of Well Fields

To be able to retrieve particle locations based on well fields (see section 14.5 PT Output Retrieval Utility (*p. 234*)) it is necessary to specify the well fields in the MIKE SHE well database file (\*.WEL).

### 14.1.3 Input to the PT Simulations

Much of the required information for the PT simulation is given in the \*.tsf input file as described in Working with the .TSF Files (*V.1 p. 196*). An example of the .tsf file that also includes the PT section is found in Transport Setup File (\*.tsf) (*V.1 p. 219*). The rest of the .tsf file includes information regarding:

- file name of water movement calculation
- Species name (note that the PT engine can only handle one species)
- Simulation title and description
- Simulation period
- Components (note that only SZ may be executed)
- Storing frequency
- Time step controls
- Porosity for groundwater (note that the dual porosity option cannot be used in PT)
- Dispersion parameters for groundwater (note that the PT engine assumes isotropic dispersion conditions)



- Location of sources (note that only SZ sources apply)
- Initial concentrations may in some cases be used to determine the initial number of particles - see below.

The \*.tsf uses the old MIKE SHE format, and a new MIKEZero GUI for AD/PT simulations has not yet been made, so at the moment the \*.tsf input file must be edited in a text editor.

| Line item                                | Comment   |
|--|---|
| <pre> ***** PARTICLE TRACKING : T </pre> | <p>The particle tracking simulation engine is activated at the bottom of the *.tsf file, i.e. below the defined bottom line by specifying this line</p>   |
| <pre> PARTICLE MASS      : 100 </pre>    | <p>The particle mass is used to convert particle counts (number of particles) to concentration.</p> <p>This parameter is not used if INITSPEC refers to number of particles.</p>  |
| <pre> INITSPEC           : -1 </pre>     | <p>This line refers to how initial concentration is specified through the parameter INITSPEC, i.e. either as number of particles or as concentration, which then is converted into number of particles from the particle mass given above.</p> <p>-1: If INITSPEC is set to -1 the initial concentration given in the MIKE SHE AD input part refers to number of particles in each cell.</p> <p>-2: If INITSPEC is set to -2 the initial concentration given in the MIKE SHE AD input part refers to “concentration”. This concentration will be converted to a number of particles by using the particle mass and the saturated volume of each of the compartments. If the resulting number of particles is not an integer a truncation to nearest integer is performed.</p> <p>+ve: If INITSPEC is specified positive it specifies the number of particles in each cell and the concentration given in the MIKE SHE AD input part is disregarded.</p> |
| <pre> VERTICAL CORRECTION : 1 </pre>     | <p>This line ensures that only a limited number of particles leaves the groundwater and moves to the overland by a correction of the vertical particle co-ordinate when moving between compartments with changing thickness.</p> <p>1: turns this option on (default)</p> <p>2: turns this option off</p>   |



| Line item                    | Comment   |
|------------------------------|---|
| CONSTANT CELL PARTICLES : F  | This line is used to avoid particles being initially located in cells with constant head or other constant concentration cell. This parameter is usually set to <i>F</i> .  |
| ONLY SATURATED : T           | This line is used to avoid particles initially being located in the unsaturated zone of the sub-surface system and thereby not being moved. This parameter is usually set to <i>T</i> .   |
| DISTRIBUTION TYPE : 3        | <p>This line determines how the particles initially are placed in each grid cell. There are three options:</p> <p>1: the particles are randomly (3D) distributed within each cell;</p> <p>2: the particles are located in a certain horizontal plane in each layer and in a rigid way. If e.g. the number of particles is specified as 2 the grid is divided into 2x2 quadratic elements and a particle is located in the centre of each of these elements. This means that 4 particles will be introduced in the model. The location of the horizontal plane is given below;</p> <p>3: the particles are located uniformly (3D) in each cell and the number of particles introduced in the model will be <math>N^3</math> where <i>N</i> is the number of particles specified by INITSPEC.</p> |
| RELATIVE INITIAL LEVEL : 0.5 | This line determines the location of the particles if the DISTRIBUTION TYPE is set to 2. The relative level is calculated from the bottom of the cell, i.e. if RELATIVE INITIAL LEVEL is set to 0.4 the particles will initially be located in a horizontal plane 40% of the cell height above the cell bottom. This applies for all layers.  |
| REGZONEFILE : none           | This line is left over from previous releases and allows the user to define registration cells via a text file with co-ordinates of the cells. Other - more relevant - options to define registration cells are given below. Specify none or a file name.   |
| DFS2 INPUT : F               | This line determines if one or more dfs2 files specifies the registration cells. Specify <i>F</i> or <i>T</i> for this parameter.   |



| Line item  | Comment  |
|--|--|
| WELLREGISTRATION : T   | If this parameter is set to T the registration cells are determined as those cells where ground-water abstraction takes place. This is a very useful option to use for calculation of capture zones. However, the user has to name the well fields in the MIKE SHE well editor to distinguish capture zones from different well fields - see PT Output Retrieval Utility (V.1 p. 234).                 |
| LPTBIN : F   | This line determines if the locations of all particles are stored for each calculation time step. Be careful: very large data files may be generated if this parameter is set to T.  |
| SPECIFY MINIMUM PTBIN OUTPUT TIME STEP : T   | This option is used to reduce the size of the PTBIN file, if the line above is True  |
| MINIMUM PTBIN OUTPUT TIME STEP HRS : 87600   | Output frequency to the ptbin file if the above option is True   |
| PT GROSS SHAPE FILE : T  | Output option to specify the particle history file as a .shp file  |
| ----- Registration codes<br>No of data elements : 2<br>Lower layer : 3<br>registration zone : upper.dfs2 1<br>Lower layer : 11<br>registration zone : lower.dfs2 1   | If DFS2 INPUT is set to T the following lines specifies the registration cells in one or more dfs2 files. The separation line is followed by the specification of the number of data elements and the lower layer and name of the dfs2 files and the item number.  |
| REGZONELENSES : T  | Lense-based registration codes. This line determines if the registration codes are defined as one or more lenses, each described by horizontal extent and upper and lower levels.<br><br><b>Note.</b> You can specify as True either DFS2 INPUT or REGZONELENSES - not both  |
| No of lenses :2<br>Grid Codes : Lense1.dfs2 1<br>Upper Level :Lense1_up.dfs2 1<br>Lower Level :Lense1_low.dfs2 1<br>Grid Codes :Lense2.dfs2 1<br>Upper Level :Lense2_up.dfs2 1<br>Lower Level :Lense2_low.dfs2 1 | If REGZONELENSES is set to True, these lines define the lenses. The Grid codes file defines the extent and registration code of the lense. The lense is present where ever there is a positive grid code.<br><br>The upper and lower levels define the top and bottom of the lenses. A global value can also be used.<br><br><b>Upper/Lower Level</b><br>EUM Data type: Elevation<br>Base Unit: metres |



## 14.2 Executing MIKE SHE PT

MIKE SHE PT is executed from a DOS prompt (similar to the MIKE SHE AD). Open the DOS prompt and navigate to the directory of interest, i.e. the directory where the transport setup file is located. The simulation is started by the command:

```
Mshe_AdvectionDispersion \\apv projectname.tsf
```

A window similar to the Water Movement simulation showing the simulation status will appear.

MIKE SHE PT may also be executed in batch mode simply by writing the above command into a file given the extension .bat and execute this file from a DOS prompt. This allows for successive simulation of various models.

## 14.3 Output from the PT simulations

The result files will be located in a folder with the same name as the \*.tsf file. Usually - but not necessarily - the same file name is used for both the WM and PT simulations. The PT result files are:

- ***projectname*.PTRES**: An ASCII file in pfs-format listing the abstraction wells and the computational cells, where abstraction occurs. Used for retrieval of particle location - see PT Output Retrieval Utility (V.1 p. 234).
- ***projectname*.PTREG** and ***projectname*.trf**: Two binary files that cannot be opened directly.
- ***projectname*.PTBIN**: An optional binary file containing all of the particle locations at every saved time step. Individual path lines can be extracted using the PTBin Output Retrieval Utility (V.1 p. 233).
- ***projectname*.PTGross.shp**: An optional point theme shape file containing the path line information of every particle at every saved time step. As part of the shape file, a .shx and a .dbf file are also created. The .dbf file can be opened in Excel if it is less than 65536 lines.
- ***projectname*\_AD\_3DSZ.dfs3**: Temporal and spatially varying SZ concentrations in the mobile phase (not relevant for PT simulations).
- ***projectname*\_PT\_3DSZ.dfs3**: Temporal and spatially varying PT results including:
  - Number of particles





- Number of registered particles
- Most recent registration zone code
- Average age
- Average transport time to nearest registration cell
- Accumulated particle count

Besides these result files the program also writes output to two log files. The error log list errors encountered during execution and the print log file contains execution step information, statistics on the run and a mass balance (if requested).

## 14.4 PTBin Output Retrieval Utility

The PTBin output retrieval program is run from a command line and requires a PFS input file. To execute the program open a command line and navigate down to the location of the working directory that contains the AD input file (\*.tsf). The program is then executed by typing:

```
PtBinRetrieval //apv projectname.pfs
```

The .pfs file required for the PTBinRetrieval utility has the following format:

| Line item                               | Comment   |
|---|---|
| [PTBinRetrieval]<br>FileVersion = 1     | File header   |
| PTBin_file name =  .\projectname.ptbin  | the path location and name of the saved PTBin file          |
| Output_file name =  .\PTBin_output.txt  | the ASCII output file where you want to save the path lines |
| NrOfParticles = 5                       | Number of particle paths required                           |
| Particle_IDs = 3000,5000,6000,7000,7200 | ID numbers of the required particles                        |
| EndSect // PTBinRetrieval               | Closing line  |

Particle IDs can be found by using the PT Output Retrieval utility.



## 14.5 PT Output Retrieval Utility

Previously, retrieval of particle locations was made by the program called “MShe\_AdvectionDispersion\_SzOr.exe” and an associated text file describing what to extract from the MIKE SHE AD result file. This approach can still be used, however, a new and easier approach has been developed, which will be described in this section. The old approach is described in the old MIKE SHE Advection-Dispersion User Guide, which can be provided on request.

The PT output retrieval utility can be used to extract ArcView shape files from particle tracking results. It allows filtering the results for:

- Destination type:
  - Specific sink types (drain, river, unsaturated zone, well, constant concentration boundary or constant concentration sink)
  - Registration codes specified by the user in the \*.tsf file
  - Wells found in the flow results
  - Well fields found in the flow results\*)
- Layer from which the particles originated
- Release (birth) time
- Transport time

Available sink codes are

- 0 - unknown
- 1 - active cell
- 2 - river sink
- 3 - drain sink
- 4 - well sink
- 5 - exchange flow to UZ (particle sink)
- 6 - constant concentration (boundary source)
- 7 - constant concentration (boundary sink)
- 9 - zero flux (boundary)
- 10 - precipitation source
- 13 - constant concentration cell (internal SZ source)

**Note:** To extract particle locations based on well fields requires that different well fields have been defined, see section Specification of Well Fields (V.1 p. 228).

The results can be written to either



- a single shape file where the point attributes allow further selection of the particles in ArcView, or
- separate files for each destination type and optionally for each layer e.g. one file for each sink type/layer combination.

The output retrieval program is run from a command line and requires a PFS input file. To execute the program open a command line and navigate down to the location of the working directory that contains the AD input file (\*.tsf). The program is then executed by typing:

```
Ptoutputretrieval //apv projectname.pt_or
```

For information on the format of the .pt\_or file, see Structure of the PT\_OR File (V.1 p. 235).

***projectname\_ptoutputretrieval.err***: If errors occur during execution of the program these are written to this log file.

#### **14.5.1 Limitations with the PT Output Retrieval**

When using registration zones to identify particles that move through certain parts of the model it should be noted that particles can appear more than once in the output. As they move from one zone to the next they are repeatedly registered and are finally also registered when they are removed from the model by a sink. An example would be a particle moving into a registration zone with code 1. The particle is then registered as being in an 'active cell' and the registration zone code and travel time to this zone is memorised. If the particle is at a later time removed by a well it will again be registered but now it will be registered as being removed by the 'Well' sink.

If there are multiple wells within one cell and output for wells is requested then the output can contain the same particle more than once. As the model does not know which of the wells the particle should be assigned to (the program looks at the total well sink for the cell and cannot distinguish individual wells) the particle will be repeated for each of the wells within the cell.

#### **14.5.2 Structure of the PT\_OR File**

The PT\_OR file is a PFS type file, which can be edited with a text editor. An example of a PT\_OR file can be found in the section, Example PT Output Retrieval File (.PT\_OR) (V.1 p. 236).

The PT\_OR file refers to codes/names found for the specific PT run. In the absence of the user interface, these codes can be found by checking in the



*projectname*.PTRES file that is generated by the particle tracking simulation. The output retrieval program 'filters' the results of a PT simulation and writes the result to shape files. The different 'filters' each have an option 'All...' which is checked first. If this option is true the program will NOT check further input for the filter. If the option is false the program will for the filters related to destination types read the number of inputs (zero or positive) and then read for each of these a section containing the appropriate filter values (sink type/registration code/ well name / well field name).

### 14.5.3 Example PT Output Retrieval File (.PT\_OR)

The following file can be found in the Examples sub-directory in your installation directory.

```
// Created      : 2004-08-6 22:51:35
// DLL id       : C:\WINDOWS\System32\pfs2000.dll
// PFS version  : Mar  3 2004 21:35:12

[PT_OR_input]
  VersionNumber = 2
  PTRESfile name = |.\aarhus_n02_sce1\aarhus_n02_sce1.PTRES|
  PTORfile name = |.\aarhus_n02_sce1\aarhus_n02_sce1|
  separatefiles = True
  separatelayers = True
  removedByWell = True
  [SpatialFilter]
    WholeModel = true
    nlaymin = 1
    nlaymax = 10
  EndSect // SpatialFilter

  [TemporalFilter]
    AllReleaseTimes = true
    ReleaseDateMin = 1990, 12, 1, 0, 0
    ReleaseDateMax = 2900, 1, 1, 0, 0
    AllTravelTimes = true
    TravelTimeMin = 0
    TravelTimeMax = 1000000000
  EndSect // TemporalFilter

  [SinkCodes_out]
    Allsinks = false
    NrofSinkCodes_out = 1
    [SinkCode]
      Code = 4
    EndSect // SinkCode

  EndSect // SinkCodes_out

  [RegCodes_out]
    AllCodes = false
```



```
NrofRegCodes_out = 0
[RegCode]
  Code = 0
EndSect  // RegCode

EndSect  // RegCodes_out
[WellNames_out]
  AllWells = false
  NrofWellNames_out = 0
  [WellName]
    Name = ''
  EndSect  // WellName

EndSect  // WellNames_out

[WellFieldNames_out]
  AllWellFields = false
  NrofWellFieldNames_out = 1
  [WellFieldName]
    Name = 'Hinnerup'
  EndSect  // WellFieldName

EndSect  // WellFieldNames_out

EndSect  // PT_OR_input
```





*Phi Software*

## **WORKING WITH TIME SERIES**







## 15 TIME SERIES DATA

MIKE SHE uses the dfs0 file format for time series data. Various tools are available for converting ASCII and EXCEL time series to the dfs0 file format. Time series data is required as input for most transient simulations, for example, daily records of precipitation. Transient simulations can also generate numerous dfs0 output files.

### 15.1 Creating Time Series in MIKE SHE

In most cases, you will create dfs0 files using the Create buttons in the MIKE SHE Setup dialogues. In this way, you can avoid the confusing task of assigning the Type of time series (e.g. precipitation) and EUM Unit type (e.g. millimetres) and the TS Type (e.g. reverse step accumulated). Each of these items are specified automatically.

If you create time a time series using a Create button, the following dialogue will appear:

**Create a new Dfs0 file**

**Contents**

- ☒ Uniform in all time intervals
- ☐ Import from old MIKE SHE T0-file format
- ☐ Import from excel file

Uniform Value:  Excel version:

Filename:

**Time series period**

Start Date:

End Date:

**Time Series Interval**

Days:  Hours:  Minutes:

**Time Series File**

Item type:

Item name:

Dfs filename:

Buttons: Create file, Cancel



The principle choice in this dialogue is whether to create an initially uniform time series file or to import a time series from an Excel file or from a file with the older .t0 file format.

### **Uniform time series**

In a uniform time series, every time step will have the same value. You should use the uniform time series option if you want to create a time series file where you do not have any data to import.

### **Time Series Period**

The time series period is the extent of the time series. In a MIKE SHE simulation, all the time series files must cover the Simulation Period (V.2 p. 28). The default time series period for a new time series file is the Simulation Period. However, if you change the time series period so that it does not cover the simulation period, you will receive an error message when MIKE SHE tries to run. If you try to add a time series file that does not cover the simulation period, then the OK button will remain greyed out and you will not be able to select the file. The constraints tab in the file selector dialogue gives you the reason that you cannot select the file.

### **Time Series Interval**

The time series interval is the length of the individual time periods. The number of time periods is the length of the time series period divided by the period interval. The last period is shortened if necessary.

### **Time Series File**

Every time series has an **Item Type** which is defined by the valid EUM Data Unit (see EUM Data Units (V.1 p. 349)) for the particular variable from which the Create dialogue was launched. In most cases, there is only one valid Type. In some cases you may have a choice. For example, in Precipitation, you can choose between Precipitation Rate, which is the average amount of precipitation per time (e.g. mm/hour) in the time interval, and Rainfall, which is the measured amount of precipitation in the time interval (e.g. mm).

The **Name** is simply the name of the data item in the resulting .dfs0 file.

The **file name** has a default value, that you should change if you will be creating several files of the same type, such as multiple rain gauge time series files. Otherwise you may accidentally overwrite the previous file.

## **15.1.1 Import from ASCII**

The easiest way to import ASCII data into a dfs0 file is via the Windows clipboard. In this case, create a uniform time series file with the correct



number of time steps and then highlight all of the data values. Then copy and paste the data from the ASCII file into the table.

However, if you want to import the data from an ASCII file, then you need to create the file from the File/New menu and choose ASCII file. This is part of the Time Series Editor itself.

### 15.1.2 Import from Excel

Only the first Excel Worksheet will be read when reading the Excel file. However, the worksheet can contain any number of columns of time series data. If there are multiple columns of data, each will be assumed to be of the same type. If the Excel file columns are of different types, then you can change the data type in the Time Series Editor.

The time series is assumed to have a non-equidistant time axis and the time series period is read from the first column of the Worksheet.

#### Worksheet Format

The first row is a header containing the names of each of the columns. Each subsequent row contains the data. The first column is the date and time (with DATE or TIME cell format), followed by the data values.

|                     | name1 | name12 | name3 |
|---------------------|-------|--------|-------|
| 01/01/1981 00:00:00 | 0.1   | 0.2    | 0.3   |
| 02/01/1981 00:00:00 | 0.304 | 0.304  | 0.304 |
| 03/01/1981 00:00:00 | 0.025 | 0.025  | 0.025 |
| 04/01/1981 00:00:00 | 0.604 | 0.604  | 0.604 |

### 15.1.3 Import from old .t0 file

The old .t0 file format is from the X-Motif version of MIKE SHE that existed before the Windows version was introduced in 2001. The .t0 file format contains all of the relevant time information. For more information on the .t0 file format, please refer to your original MIKE SHE documentation.

## 15.2 Working with Spatial Time Series

In the MIKE SHE Toolbox, there is a Tool in the File Converter section called *dfs2+dfs0 to dfs2*. In this utility you specify a dfs2 grid file with integer grid codes and a dfs0 file with time series data, where the dfs2 file grid codes are the item numbers in the dfs0 file.



The utility will read the dfs2 file and for each time step in the dfs0 file, it will substitute the grid code with the time series value.

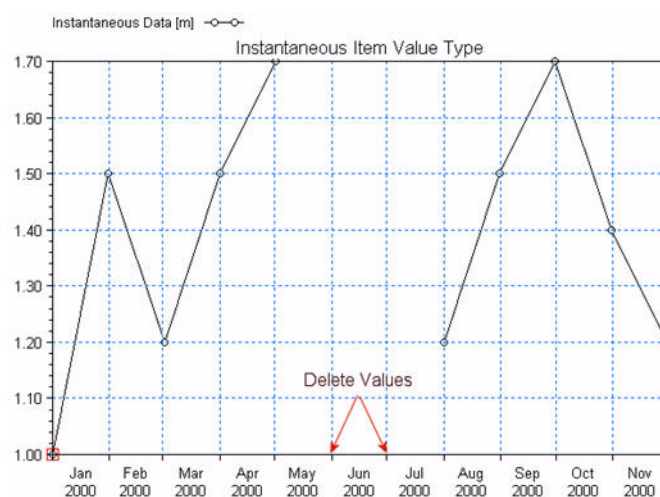
The result is a dfs2 file with one grid for each time step and the grid values are the time series values.

## 15.3 Time Series Types

Specifies how the time step is being defined and how the measured value is being assigned to the time step. There are five different value types available:

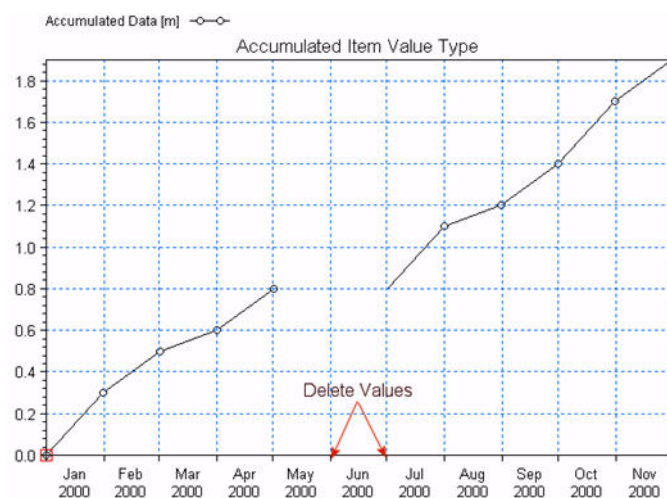
### Instantaneous

The values are measured at a precise instant. For example, the air temperature at a particular time is an instantaneous value.



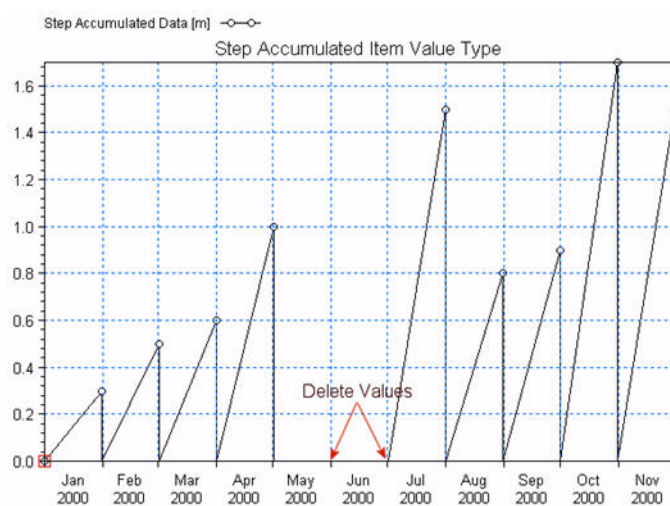
### Accumulated

The values are summed over successive intervals of time and always relative to the same starting time. For example, rainfall accumulated over a year with monthly rainfall values.



### Step Accumulated

The values are accumulated over a time interval, relative to the beginning of the interval. For example, a tipping bucket rain gauge measures step-accumulated rainfall. In this case, the rain gauge accumulates rainfall until the gauge is full, then it empties and starts accumulating again. Thus, the time series consists of the total amount of rainfall accumulated in each time period - say in mm of rainfall.

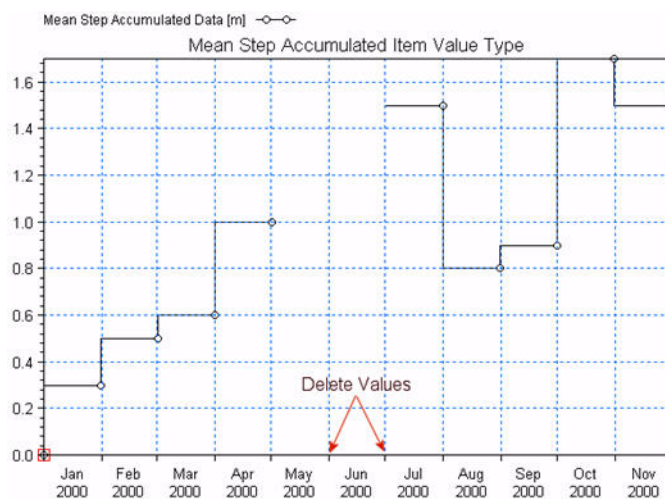


### Mean Step Accumulated

The values are accumulated over the time interval as in the Step Accumulated, but the value is divided by the length of the accumulation period. Thus, based on the previous example, the time series consists of the rate of

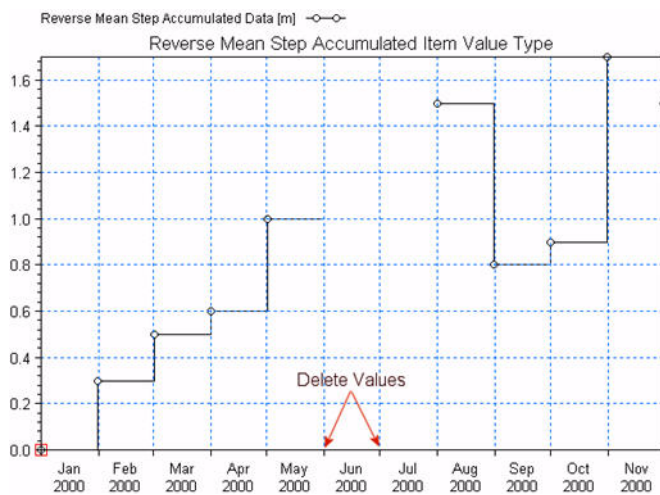


rainfall accumulated in each time period - say in mm of rainfall per hour (mm/hr).



### Reverse Mean Step Accumulated

In this case, the values are the same as the Mean Step Accumulated, but the values represent the time interval from now to the start of the next time interval. The Reverse Mean Step Accumulated time series are primarily used for forecasting purposes.





*Phi Software*

**TIME SERIES EDITOR**







---

## 16 INTRODUCTION

The appearance of the Time Series Editor differs if you create a new (blank) time series compared to opening an existing data (\*.dfs0) file.

Creating a new time series requires specification of properties for the time series file, and the File Properties dialog is therefore opened in this case.

If you are opening an existing time series data file, the data are immediately presented in the Time Series data dialog where data can be viewed and edited both in a graphical and in a tabular view. In this case, if you wish to change the already defined file properties, it is required to open the File Properties dialog from the graphical view.

You operate the Time Series Editor from the main menu, the tool bar icons, or by right-clicking on the graphical view.



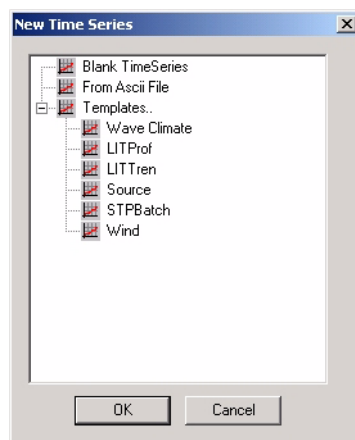


## 17 **PROPERTIES**

### 17.1 **New File Dialog**

This dialog is used to create a new Time Series.

It is possible to create a "Blank" data set, import from ASCII file or select from a number of pre-defined templates containing different sets of properties.



If you chose "Blank Time Series", the File Properties Dialog is displayed with a set of default properties. You can then customize the time series according to your own needs.

If you choose From ASCII File, the import from ascii Dialog is displayed where you can set the properties to import from ascii.

If you choose one of the template files, the File Properties Dialog is displayed with a set of properties specific to this template. It may not be possible to edit all of the properties. The following templates are available:

- Wave Climate Template
- LITProf template
- LITTren template
- Source Template
- STPBatch template
- Wind template



### 17.1.1 Wave Climate Template

Wave Climate template creates a timeseries with the following pre-defined properties

- Equidistant Calendar Axis
- 10 seconds timestep
- 10 Timesteps
- 15 Items:
  - Duration (pct. year) of type Undefined, unit percent and TS Type Step Accumulated
  - Wave Height of type Wave height, unit meter and no TS Type defined
  - Wave Direction of type Wave direction, unit degree and no TS Type defined
  - Profile Number of type Profile number, unit () and no TS Type defined
  - Wave Period of type Wave period, unit second and no TS Type defined
  - Ref. Depth, height of type Water Level, unit meter and TS Type Instantaneous
  - Ref. Depth, angle of type Water Level, unit meter and no TS Type defined
  - Mean water level of type Water Level, unit meter and no TS Type defined
  - Spectral Description of type Spectral description, unit () and no TS Type defined
  - Spreading Factor of type Spreading factor, unit () and no TS Type defined
  - Current Speed of type Flow velocity, unit m/s and no TS Type defined
  - Ref. No for current of type Reference point number, unit () and no TS Type defined
  - Wind Speed of type Wind Velocity, unit m/s and no TS Type defined
  - Wind Direction of type Wind Direction, unit degree and no TS Type defined



- Wind Friction coeff. of type Wind friction factor, unit () and no TS Type defined

The Title is Wave Climate and Start time is 01-01-2000 00:00:00. You can only edit the title, the time step and the number of time steps.

It is always possible to customize the data set. When the data set has been created, you can right-click in the graphical view and select "Properties". The File Properties Dialog is displayed, and all the properties can be edited.

### 17.1.2 *LITProf template*

LITProf template creates a timeseries with the following pre-defined properties

- Equidistant Calendar Axis
- 1 hour timestep
- 10 Timesteps
- 6 Items:
  - Time of type Undefined, unit hour and no TS Type defined
  - Wave Height of type Wave height, unit meter and no TS Type defined
  - Wave Direction of type Wave direction, unit degree and no TS Type defined
  - Wave Period of type Wave period, unit second and no TS Type defined
  - Spreading Factor of type Spreading factor, unit () and no TS Type defined
  - Water Level of type Spreading factor, unit () and no TS Type defined

The Title is LITProf and Start time is 01-01-2000 00:00:00. You can only edit the title and the number of time steps.

It is always possible to customize the data set. When the data set has been created, you can right-click in the graphical view and select "Properties". The File Properties Dialog is displayed, and all the properties can be edited.

### 17.1.3 *LITTren template*

LITTren template creates a timeseries with the following pre-defined properties



- Equidistant Calendar Axis
- 1 hour timestep
- 10 Timesteps
- 7 Items:
  - Time of type Undefined, unit hour and TS Type Instantaneous
  - Wave Height of type Wave height, unit meter and no TS Type defined
  - Wave Angle of type Wave direction, unit degree and no TS Type defined
  - Wave Period of type Wave period, unit second and no TS Type defined
  - Current Speed of type Flow velocity, unit m/s and no TS Type defined
  - Current Angle of type Flow direction, unit degree and no TS Type defined
  - Mean water level of type Water Level, unit meter and no TS Type defined

The Title is LITTren and Start time is 01-01-2000 00:00:00. You can only edit the title and the number of time steps.

It is always possible to customize the data set. When the data set has been created, you can right-click in the graphical view and select "Properties". The File Properties Dialog is displayed, and all the properties can be edited.

#### **17.1.4 Source Template**

Source template creates a timeseries with the following pre-defined properties

- Equidistant Calendar Axis
- 1 hour timestep
- 10 Timesteps
- 15 Items:
  - Duration (pct. year) of type Undefined, unit percent and TS Type Instantaneous
  - Item 2 to 15 - Source Value of type Undefined, unit  $m^3/s$  and no TS Type defined



The Title is Source and Start time is 01-01-2000 00:00:00. You can only edit the title and the number of time steps.

It is always possible to customize the data set. When the data set has been created, you can right-click in the graphical view and select "Properties". The File Properties Dialog is displayed, and all the properties can be edited.

#### 17.1.5 STPBatch template

STPBatch template creates a timeseries with the following pre-defined properties

- Equidistant Calendar Axis
- 10 seconds timestep
- 10 Timesteps
- 13 Items:
  - Water Depth of type Water Level, unit meter and no TS Type defined
  - Wave Height of type Wave height, unit meter and no TS Type defined
  - Wave Period of type Wave period, unit second and TS Type Instantaneous
  - Wave Direction of type Wave direction, unit degree and TS Type Instantaneous
  - Breaking Wave (0/1) of type Undefined, unit undefined and no TS Type defined
  - Current Speed of type Flow velocity, unit m/s and no TS Type defined
  - Current Direction of type Flow direction, unit degree and no TS Type defined
  - Mean Grain Size of type Grain diameter, unit millimetre and no TS Type defined
  - Sediment Spreading of type Geometrical deviation, unit () and no TS Type defined
  - Water temp. of type Temperature, unit degree Celsius and no TS Type defined
  - Bed Slope x-dir (m/m) of type Bed slope, unit () and no TS Type defined



- Bed Slope y-dir (m/m) of type Bed slope, unit ( ) and no TS Type defined
- Duration (pct. year) of type Undefined, unit percent and no TS Type defined

The Title is STPBatch and Start time is 01-01-2000 00:00:00. You can only edit the title, the time step and the number of time steps.

It is always possible to customize the data set. When the data set has been created, you can right-click in the graphical view and select "Properties". The File Properties Dialog is displayed, and all the properties can be edited.

### **17.1.6 Wind template**

Wind template creates a timeseries with the following pre-defined properties

- Equidistant Calendar Axis
- 10 seconds timestep
- 10 Timesteps
- 2 Items:
  - Speed of type Wind velocity, unit m/s and no TS Type defined
  - Direction of type Wind direction, unit degree and TS Type Instantaneous

The Title is Wave Climate and Start time is 01-01-1990 00:00:00. You can only edit the title, the time step and the number of time steps.

It is always possible to customize the data set. When the data set has been created, you can right-click in the graphical view and select "Properties". The File Properties Dialog is displayed, and all the properties can be edited.

## **17.2 File Properties Dialog**

This dialog is used to view and change the properties of the time series being visualized.





**File Properties**

**General Information**

Title:

**Axis Information**

Axis Type:

Start Time:

Time Step:  [days]  
 [hour:min:sec]  
 [fraction of sec.]

No. of Timesteps:  Axis Units:

**Item Information**

|   | Name     | Type      | Unit      |       |
|---|----------|-----------|-----------|-------|
| 1 | Untitled | Undefined | undefined | Inste |

Insert Append Delete

### 17.2.1 General Information

#### Title :

The title for the data contained in the file. Any text can be typed here

### 17.2.2 Axis Information

#### Axis Type :

The type of the time axis. You can select between the following types:

- **Equidistant Calendar Axis** : Data is stored with a fixed time interval and start at an absolute date and time.
- **Equidistant Relative Axis** : Data is stored with a fixed time interval, but do not have a absolute start date and time. The start time is not applicable in this case.
- **Non-Equidistant Calendar Axis** : Each data point is stored at a specific arbitrary absolute time. The time step is not applicable in this case.
- **Non-Equidistant Relative Axis** : Each data point is stored at a specific arbitrary relative time. The start time and time step are not applicable in this case.



- **Relative Item Axis :** Non-time varying data. The first item is used for the X axis and the succeeding items are plotted against this item. The start time and the time step are not applicable in this case. You can also specify the unit for the X axis.

**Start Time :**

The start time of the data. This is only relevant for calendar axis data. The format used is the standard windows format. To change this edit the regional settings in the windows control panel.

**Time Step :**

The timestep for the data. Only relevant when the time axis is equidistant (Equidistant Calendar Axis or Equidistant Time Axis). You can specify days, hour:minute:second and milliseconds. A timestep of one hour would thus be given as “01:00:00” in the [hour:min:sec] input box.

**No. of Timesteps :**

Number of time steps. If this number is changed, time steps are added or removed as appropriate at the end of the time series. When adding timesteps, the new timesteps added will be filled with an empty value, meaning that no value has been inserted.

### 17.2.3 Item Information

**Name :**

Text that identifies the item.

**Type :**

The type of the data contained in the item, indicating if it is e.g. a water level, wave height, etc. It is possible to select from a number of types using the combo box which appears if you click in the field. If a type not contained in the list is needed, write the type in the text field. This also applies to the unit below.

**Unit :**

Text that identifies the unit of the item. Unit is always related to the type. It is possible to select from a number of units using the combo box which appears if you click in the field. If a unit not contained in the list is needed, write the unit in the text field.

**TSType :**

The Item type of data. It's used to specify the meaning of the data values. You can select between the following types:



- **Instantaneous** : means that the values are representative at one precise instant. For example, the wind velocity is an instantaneous value
- **Accumulated** : means that the values are representative of one successive accumulation over the time and always relative to the start of the event to register values from. For example, the rainfall accumulated over the year if we have monthly rainfall values.
- **Step Accumulated** : means that values are representative of an accumulation over a timestep. For example, rainfall is a step-accumulated value in the following context. Let's say that we start measuring rainfall at 10:00:00. At 11:00:00 someone picks the recipient where rain fell, registers the value of 10 and empties the recipient. At 12:00:00 the same process but with a value of 15 and so on. So, in a timeseries we shall have the value 10 at timestep 11:00:00 and the value 15 at timestep 12:00:00 and so on. Values represent the timespan between the previous timestep and the current timestep
- **Mean Step Accumulated** : means that values are representative of an average accumulation per timestep. Picking the previous example, a mean step accumulated type could be used to register the average rainfall per 15 minutes. Which would be 2.5 (we are working now with 15 minutes and registering in 1 hour timestep) for 11:00:00 and 3.75 for 12:00:00 and so on. Values represent the timespan between the previous timestep and the current timestep
- **Reverse Mean Step Accumulated** : is equal to Mean Step Accumulated type, but values represent the timespan between the current timestep and the next Timestep. Used for forecasting purposes

The representation for each of the TS Types is different, since the physical meaning of values is also different. Please refer to TS Types graphical representation.

**Min. :**

Minimum value for all data in the item. This value is not editable since it's based on statistical information of the Item.

**Max. :**

Maximum value for all data in the item. This value is not editable since it's based on statistical information of the Item.

**Mean :**

Mean value for the data in the item. This value is not editable since it's based on statistical information of the Item.

**Insert :**

Insert an item before the selected one. This item will be named “Untitled” and you can immediately edit the Item properties.

**Append :**

Append an item at the end of the list. This item will be named “Untitled” and you can immediately edit the Item properties.

**Delete :**

Delete the selected item. You cannot delete an Item if it is the last one in the list, but trying to delete it causes that all properties in the Item will be cleared.

### **17.3 *Tabular View***

This view shows the data in a tabular form.

You can select entire rows or columns by clicking on one of the grey cells. Data can be cut and pasted freely.

The time column is greyed out for equidistant axis, as editing the time has no meaning in that case.

The time is shown in the default windows format selected on your computer. You can change this by editing the regional settings in the windows control panel.

You can move around in the table by using the arrow buttons or the TAB or ENTER keys. If the TAB and ENTER key is pressed at the right most column, the active cell is moved to the first column in the next line. This can be used to quickly enter data in a typewriter fashion. SHIFT+TAB or SHIFT+ENTER works the other way. If you are at the bottom right cell of the table and press TAB or ENTER, a new row is added. The time is extrapolated from the previous values and the item values are empty.

The currently selected cell can also be seen in the Graphical View as a square around the value that corresponds to it.

### **17.4 *Graphical View***

This view presents the data graphically.



By pressing the right mouse button a pop-up menu is displayed. This menu can be used to zoom, enable options, customize the representation, select sub-sets or select which items are shown.

You can know the precise point where the mouse pointer is positioned looking at the Status Bar at the bottom of the screen.



### 17.4.1 Zoom

You can zoom in and out on the data shown, use previous zoom, use next zoom or refresh the view using the Zoom In, Zoom Out, Previous Zoom, Next Zoom and Refresh commands accessible from the menu that pops up right clicking on the view, from the View menu or even from the Zoom toolbar



The first icon represents the Zoom In command, the second one the Zoom Out, The third one the Previous Zoom, the fourth one the Next Zoom and the fifth one enables or disables the grid lines in the view, which is also accessible from the menu that pops up right clicking on the view.

When you zoom in, scroll bars are displayed at the bottom and right hand side of the view. You can pan by moving the scroll bars.

### 17.4.2 Editing modes

Data can be edited graphically by using four modes:

- **Select points** : allows you to select points. When clicking in a point that point is selected. A red square around the point appears and the correspondent cell in the Tabular View is selected
- **Move points**: allows you to move points. When the mouse pointer is near a point, the pointer becomes a moving cross and you can move the point by moving the mouse pointer while keeping the left button of the mouse down (drag)
- **Insert points** : allows you to insert new points in the data set just by positioning the mouse pointer where the new point shall be located and click the left button. When this mode is enabled, the mouse pointer becomes a pencil. It's only possible to select this mode in a Non Equidistant Axis type.



- Delete points : allows you to delete points. When this mode is selected, when the mouse pointer is near a point, the pointer becomes a rubber and clicking on the left button of the mouse, deletes the point. The point is not deleted from the data but its value is set to empty.

You can select these four modes through the pop-up menu, the Edit menu or using the Mode Toolbar



The first icon enables the Select Points Mode, the second one enables the Move Points Mode, the third one enables the Insert Points Mode and the fourth one enables the Delete Points Mode.

When a file is opened, all items contained in the file are by default plotted. The title contained in the file is used as an header and the item names are displayed in the upper left hand corner of the plot.

If data are associated with a calendar axis, the date and hour is plotted at appropriate intervals. If data are associated with a relative axis, a normal X axis is shown.

The axes are scaled automatically so all data presented are shown.

### 17.4.3 Graphical and font settings

The appearance of the text and graphics can be controlled through the graphics and font commands in the Settings menu or the pop-up menu.

In the graphics settings, you can select which point style (or no point at all) you want to use for each item, which line style (or no line at all) you want to use to connect the points and the text format to apply to the points labels (if desired).

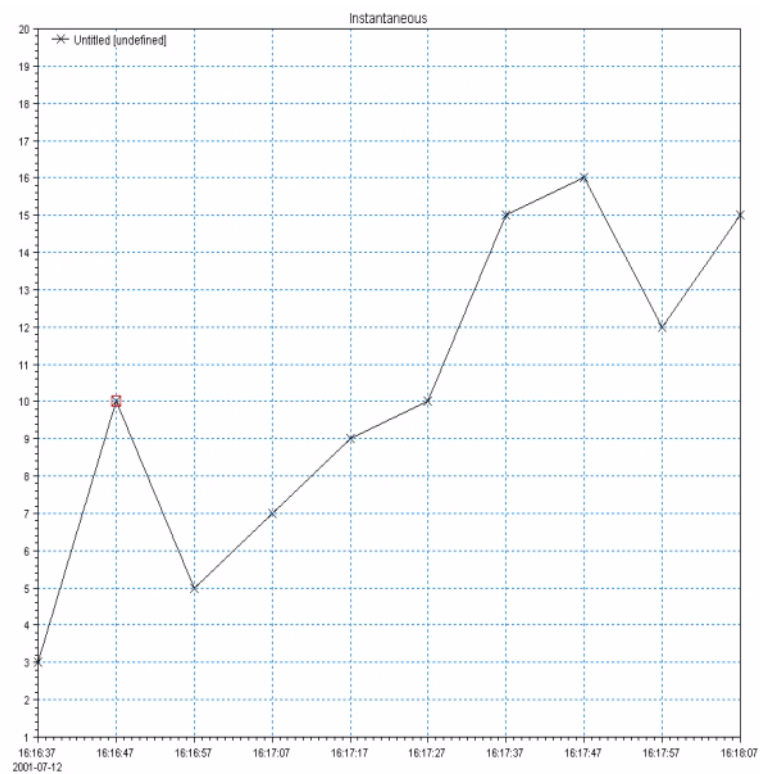
In the font settings, you can select the font, font style, size, font effects and font color to use in the legends.

Please refer to Graphical Settings Dialog and Font Settings Dialog

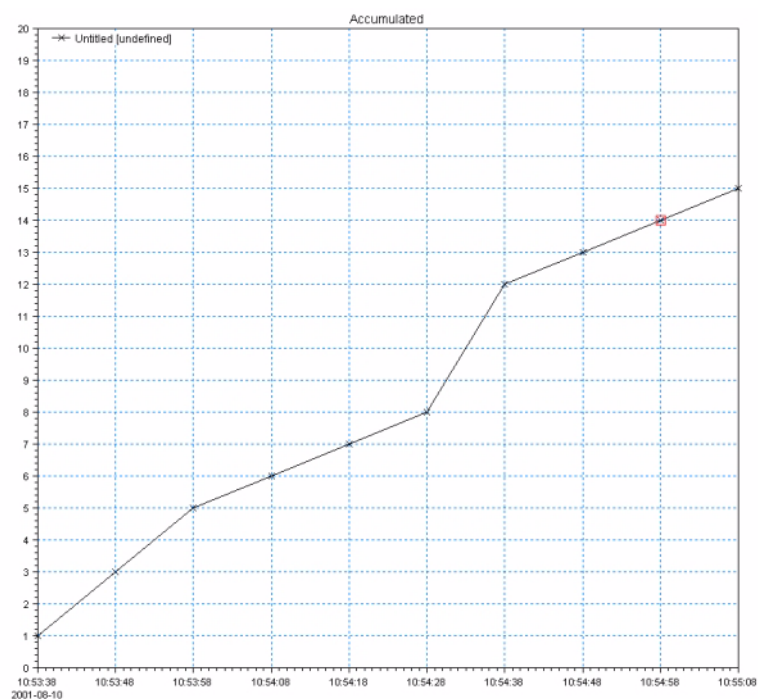
### 17.4.4 TS Types graphical representation

The representation of the data depends on the TS Type of the items.

- Instantaneous : Points are connected by lines. Empty data (delete values) are marked at the x-axis.

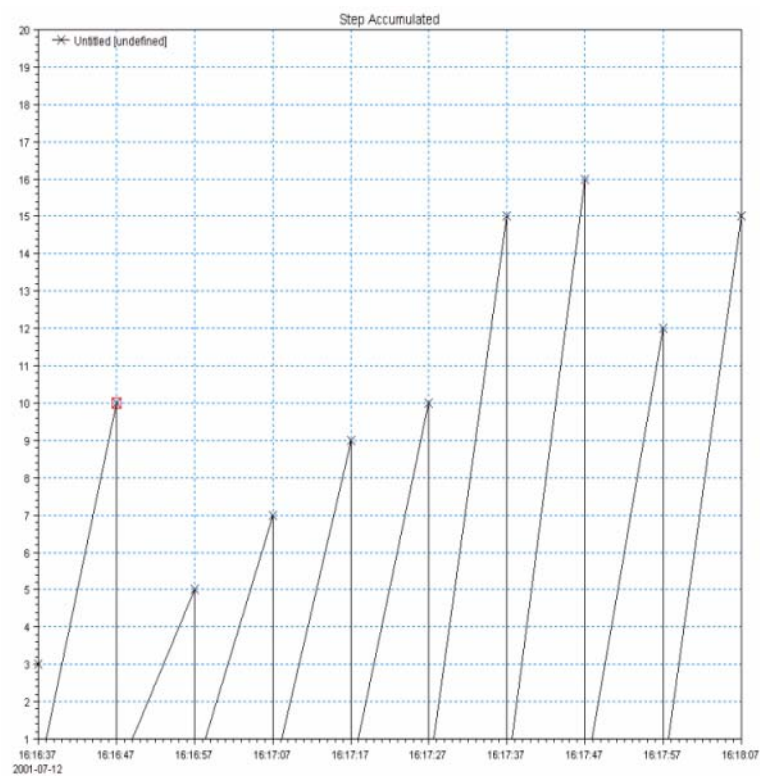


- Accumulated :The same as Instantaneous. However, an Accumulated timeseries shall be always and increasing line.

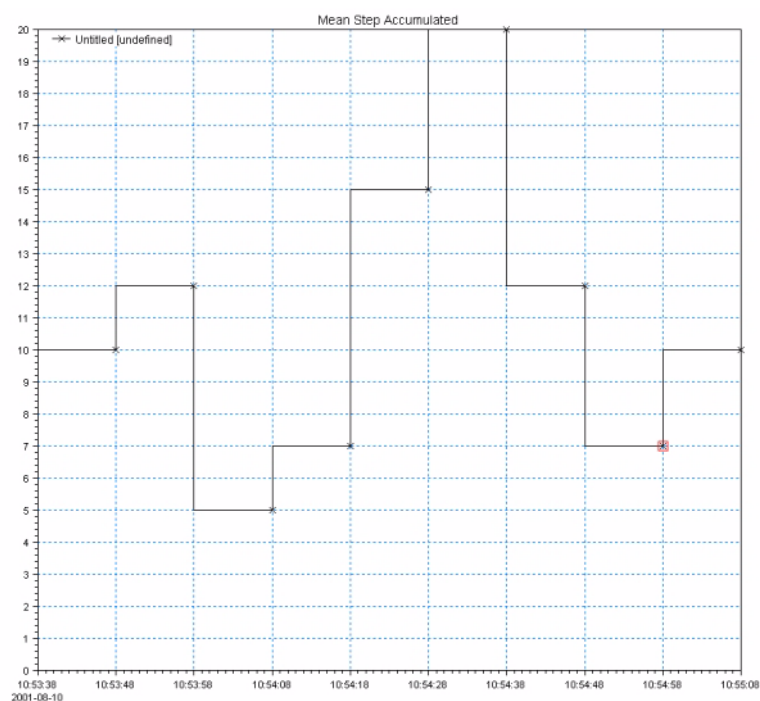


- Step Accumulated : A line is drawn from the x-axis previous timestep till the point.

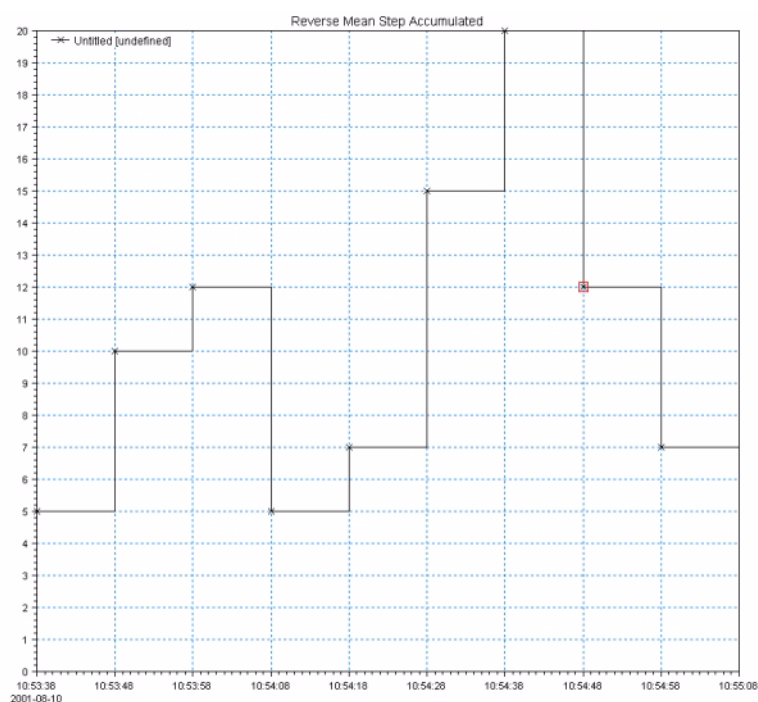




- Mean Step Accumulated : A line is drawn from the previous timestep till the current timestep with the value at current timestep

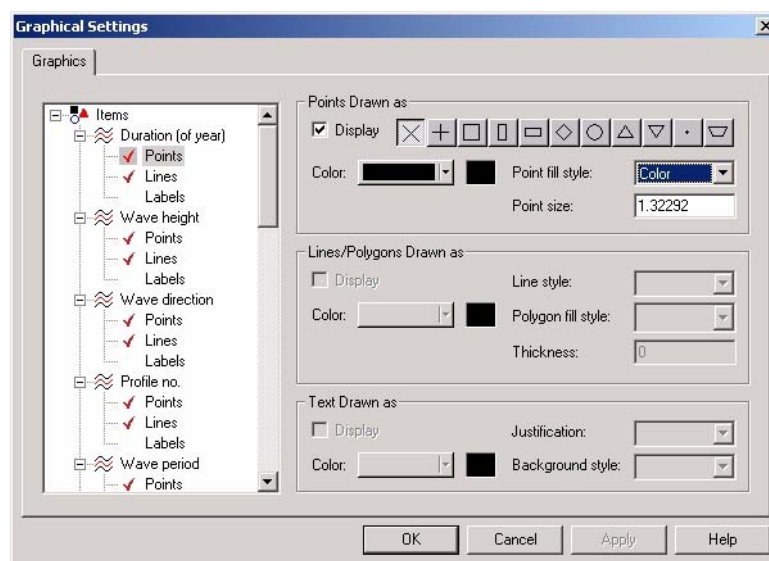


- Reverse Mean Step Accumulated : A line is drawn from the current timestep till the next timestep with the value at current timestep



## 17.5 Graphical Settings Dialog

This dialog is used to change the settings of the graphical view.



On the left hand side, the dialog shows the items organized in a tree structure. Each item has branches for points, lines and labels. By selecting a branch it's settings can be changed in the right hand side of the dialog.

For points, you can select the point mark, the point mark color, the point mark fill style and the point mark size and you can also enable/disable the point marks.

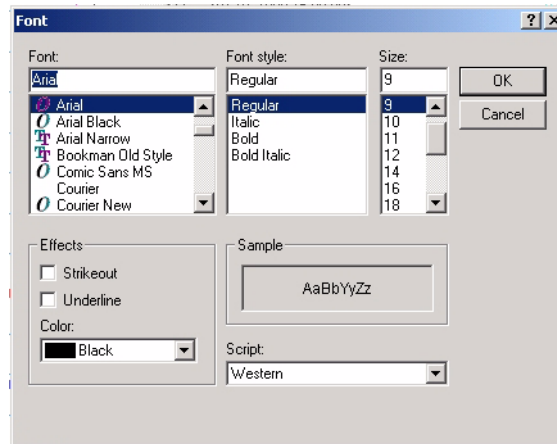
For lines, you can select the line style, the line color, the line fill style and the line thickness and you can also enable/disable lines.

For labels, you can select the text justification, the text color, the text background style and you can also enable/disable labels.

It is also possible to control if items are displayed or not by using the right mouse button on top of the item name in the tree structure.

## 17.6 Font Settings Dialog

This dialog is used to change the settings of the font to use in the graphical view.



You can select the font, font style, font size, font effect like strike out, underline and color and the script (language resource) to use for special characters.

## 17.7 File Formats

The Time Series Editor supports two file formats, the DFS Format and ASCII.

Note that files can be converted from one format to the other, i.e. saved in a different format than they opened in, but with restrictions (see below).

### DFS :

This format is developed by DHI for storage of hydrodynamic time varying data. Both zero, one, two and three dimensional data can be stored, although only zero dimensional data is relevant for the time series editor.

Files saved in this format must have the extension .dfs0 or .dfs. dt0 is also allowed, since it is the old timeseries file format.

### ASCII :

This is a generic text format which can be produced by almost any spreadsheet or text editors. Only non-equidistant calendar axis data can be saved in this format. Files must have the following format :

Title

Time Itemname 1 Itemname 2

Unit 100182 1003 2 100256 1800 1



1996-12-24 18:00:00 1.23 2.34

1996-12-24 18:30:00 1.44 3.38

1996-12-24 19:00:00 2.12 4.63

etc...

The first line contains the title.

The second line contains the string "Time" followed by the name of the items. The list is separated with tabs.

The third line is optional. It contains the string "Unit" followed by three values per item specifying unit item type, unit type, and time series values type, usually as a result of a previous export.

Each of the following lines contain data for one time step. Each line consists of a date and time followed by one field for each of the data items.

The date and time format follows the ISO standard 8601, which is YYYY-MM-DD HH:MM:SS. Between the date and time there can be a 'Space' or the letter 'T'. Following the time there must be a Tab and each of the data items must be separated by Tabs.

Note, that the date and time format shown in the example above is not the same as in the tabular view, and therefore you cannot paste the example data into that view.

Files saved in this format can have any extension except .dfs0 and .dfs.

## **17.8 Import from ASCII File**

This functionality can be used to import time series data from an ASCII file. The data set can then be saved as a dfs file or exported to an ASCII file again. Please refer to File Formats.

To import time series data, go to 'File', 'New' and select 'Time Series' under the MIKE Zero heading. This will open the 'New Time Series' dialog. Choose 'From Ascii File', and press 'OK'.



**Time Series Editor: Import from ascii**

Description

File name:

Delimiter:  Time description:

☐ Treat consecutive delimiters as one

☐ Ignore delimiters in beginning of line

☒ Deliminator between time and first item

Start Time:

Time Step:  [days]

[hour:min:sec]

[fraction of sec.]

Delete value:

☒ Time Series Export ASCII Format

Preview

|   |  |
|---|--|
|   |  |
| 1 |  |
| 2 |  |
| 3 |  |
| 4 |  |
| 5 |  |
| 6 |  |

OK Cancel

### File to Import

On the Import from ascii dialog select the ASCII file from which you wish to import the data. The ASCII file must have a certain format in order to be read correctly, see File Formats.

### Delimiter

Choose the Delimiter that separates the data in the ASCII File. When you use Timeseries Editor to export to ASCII, the TAB is used as delimiter.

### Time description

Choose the axis type of the data in the ASCII file. It's impossible to know which axis type the data in the ASCII file has. So, the user interaction is needed in this property. You can select all the axis types: Equidistant Calendar Axis, Equidistant Relative Axis, Non-Equidistant Calendar Axis, Non-Equidistant Relative Axis and Relative Item Axis. Please refer to Axis Information

**Treat consecutive delimiters as one**

Set this option active means that all consecutive delimiters are treated as one, e.g. ,if the delimiter is a TAB and there are 5 consecutive TABs, the import from ASCII will deal with these 5 TABs as only one.

**Ignore delimiters in beginning of line**

All delimiters in beginning of lines are ignores when this option is activated.

**Delimiter between time and first item**

When this option is activated, there must be a delimiter between the time for each timestep and the first item value. Otherwise, the time for each timestep is just followed by the first item value.

**Delete Value**

Fill in the Delete Value used in the file. The Delete value should be a number not typical for the data and that represents meaningless data. When a delete value is found on the data, the correspondent cell in the tabular view is empty and there is no point in the graphical view.

**Time Series Export ASCII format**

When this option is enabled, all the settings are set according to the format a timeseries is exported to ASCII using Timeseries Editor. You can only disable this option if you set an Equidistant axis type. Doing so, you can than set the Start Time, the timestep and the number of timesteps supposed to exist in the ASCII file.

**File preview**

Just bellow the description properties, there is the file preview, where you can see the top part of the ASCII file specified.

**Import preview**

Below file preview, there is the import preview. Here you can preview the result of the import with the selected description properties and change the properties till you get the expected result

After all description properties are set as wished, click the OK button and the import is done.





## 17.9 *Export to ASCII File*

You can Export a Timeseries to an ASCII File. For further description of Time Series File Formats, see File Formats.

Go to 'File' and 'Export to ASCII'. In the pop-up window you can specify where the ASCII File should be saved and under which File Name.

The file is exported using default Timeseries Editor properties. When importing an ASCII file exported by Timeseries Editor, just activate the Time Series Export ASCII format property on the Import from ASCII dialog and the import will be made with the expected result. Please refer to Import from ASCII File



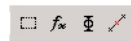


## 18 TOOLS

This is a set of tools available to work with the timeseries data.

- Calculator can be applied for several calculations on the data.
- Interpolation can interpolate missing values (delete values) in the data.
- You can select a sub-set of the data to work with using the Select Sub-Set tool.
- You can see statistical information of the data using the Statistics tool.

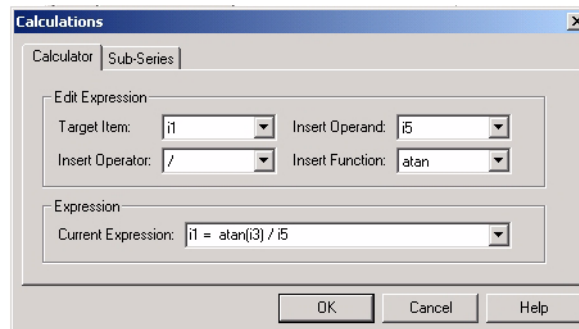
Tools are accessible through the Tools menu or by clicking on the desired tool icon on the Tools Toolbar:



### 18.1 Calculator

Calculator tool is used to set item's data based on calculations.

You can access Calculator tool from Calculator in the Tools menu or by clicking the Calculator icon in the Tools Toolbar (the second icon in the Tools Toolbar).



Here you can specify the calculation to make and which Item to set the calculated data in.

Target Item specified which item will have the data set based on the calculation specified. Item is specified by 'i' followed by the number of the item sequence in the timeseries.



You can insert a Operand, which basically, is an item. the value of the Operand item is then used to make the calculations of each of the timesteps (all timesteps in the item or all the timesteps in the current Sub-Set that you can specify).

You can also insert an Operand which basically is + for addition, - for subtraction, \* for multiplication and / for division as well as a mathematical function from the functions list.

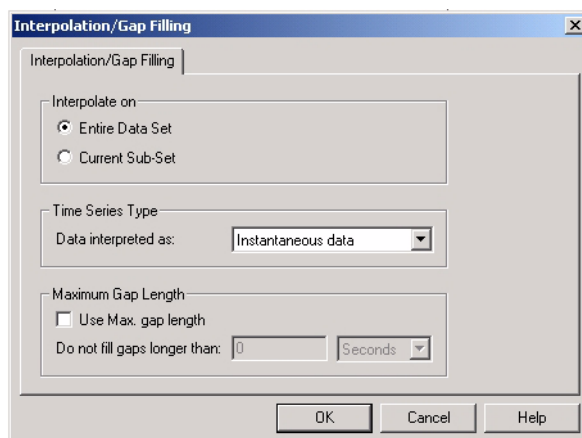
As an example, if you select the target item as item 2, then insert item 3 as Operand, then insert Operator +, then insert item 4 as Operand, then insert Operator /, then insert function cos and finally insert item 1 as Operand, you should get the final expression of  $i2 = i3 + i4 / \cos(i1)$ .

You can also specify a Sub-Set where the calculation will be made, using the tab Sub-Series that appears when you select Current Sub-Set. Please refer to Select Sub-Set.

## 18.2 Interpolation

Interpolation tool is used to interpolate missing values (delete values).

You can access Interpolation tool from Interpolation in the Tools menu or by clicking the Interpolation icon in the Tools Toolbar (the fourth icon in the Tools Toolbar).



You can choose where to interpolate. If you select Entire Data-Set, the interpolation will be done in the entire data of the currently selected item (the one that corresponds to the current cell selected in the Tabular View).



If you select Current Sub-Set, the interpolation will be done in the Current Sub-Set. You can also select at this moment the Sub-Set to use, using the tab Sub-Series that appears when you select Current Sub-Set. Please refer to Select Sub-Set.

You can also specify which items to interpolate, using the Item Range tab that appears when Current Sub-Set is enable in the Interpolation Dialog. Please refer to Select Sub-Set.

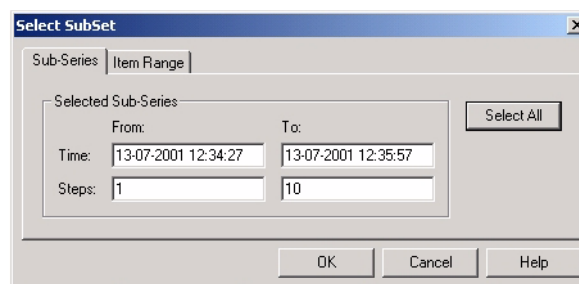
In the Interpolation tab you can also select how the data is interpreted. Since each TS Type has a different physical meaning, the interpolation is handled in a different way for all the 4 types. Only Instantaneous, Accumulated, Step Accumulated and Mean Step Accumulated TS Types are supported.

You can also specify the maximum allowed gaps (missing values) so that interpolation is done. Activating Use Max. Gap length you can specify the maximum allowed gap duration. If a gap bigger than the length specified is found, interpolation will not be done for this timestep(s).

### 18.3 Select Sub-Set

Select sub-Set is used to specify a sub-Set of the data to work with.

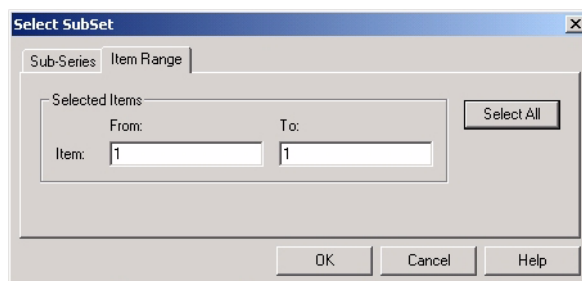
You can access Select sub-Set tool from Select Sub-Set in the Tools menu, by clicking the Select Sub-Set icon in the Tools Toolbar (the first icon in the Tools Toolbar) or by selecting Select Sub-Set from the menu that pops up right clicking on the Graphical View.



Here you can specify the time where the Sub-Set begins and the time where the Sub-Set ends or, alternatively, the timestep where the Sub-Set begins and the timestep where the Sub-Set ends. Clicking on the Select All button, selects the entire data set.



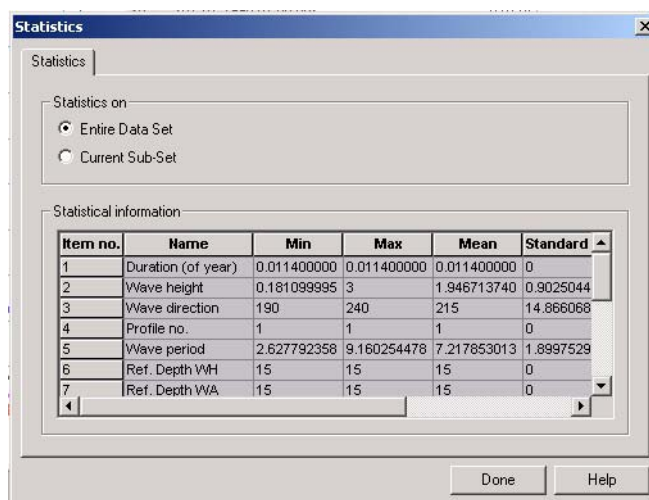
Clicking on the tab Item Range you can also specify which items belong to the Sub-Set.



## 18.4 Statistics

Statistics tool is used to view statistical information for all the items in the timeseries data set.

You can access Statistics tool from Statistics in the Tools menu or by clicking the Statistics icon in the Tools Toolbar (the third icon in the Tools Toolbar).



On the Statistics dialog, you can see the all item Names, all item Minimum values, all item Maximum values, all item Mean values, all item Standard Deviation values and the number of missing values (delete values) for the Entire Data-Set.



You can also specify use Sub-Set clicking on the Current Sub-Set. To specify a different Sub-Set, please refer to Select Sub-Set.







*Phi Software*

## **WORKING WITH SPATIAL DATA**





## 19 SPATIAL DATA IN MIKE SHE

Spatial data includes all model data that can be location dependent, for example precipitation rates and soil parameters.

There are two basic types of spatial data in MIKE SHE - Real and Integer. Real data is generally used to define model parameters, such as hydraulic conductivity. Integer data is generally used to define parameter zones. Thus, model cells with the same integer value can be associated with a time series or other characteristic.

Furthermore, real spatial parameters can be distinguished by whether or not they vary in time. At the moment Integer zones cannot vary with time.

Thus, spatial parameters can be divided into the following:

- Stationary Real Parameters
- Time Varying Real Parameters, and
- Integer Grid Codes

### Stationary Real Parameters

Stationary Real Parameters can vary spatially but do not usually vary during the simulation, such as hydraulic conductivity. If such parameters do vary in time, then you must divide the simulation into time periods and run the each time period as a separate simulation, starting each simulation from the end of the previous simulation. This is most easily accomplished using the Hot Start facility, which is found in the Simulation Period dialogue.

The spatial distribution of stationary real parameters are entered using the Stationary Real Data dialogue

### Time Varying Real Parameters

Many spatial parameters are time dependent, such as precipitation rate. In this case, both a spatial distribution, as well as a time series for each cell in the model, must be defined. Spatially distributed parameters that also vary in time are entered using the Time-varying Real Data dialogues

## 19.1 Using MIKE SHE with ArcGIS

MIKE SHE has been designed to work smoothly with ArcGIS files. In most cases, distributed data can be linked directly to shape files created by ArcGIS or any other application. The type of shape file depends on the



type of data. Distributed data, such as initial water levels can be input as point and line themes, whereas spatial data that is referenced to a time series, such as precipitation, can be added as a polygon theme. In this case, each polygon can be assigned a time series of values.

In the reverse direction, all gridded data in the MIKE SHE Setup Editor can be easily saved as a point theme shape file from the pop-menu when you right click on a colour shaded map. This includes both interpolated data in the Setup tab and pre-processed data in the Pre-processed tab.

ArcGIS grids yet cannot be added directly in the MIKE SHE Setup Editor, but they can be converted to the dfs2 file format. Select New, then the MIKE Zero Tool box and choose GIS in the list. The Grid2Mike tool will convert your ArcGIS grid files to the dfs2 file format. Support for native ArcGIS grid files will be available in a service pack later this year.

The MIKE Zero Tool box also contains tools for converting dfs2 files to ArcGIS shape files (Mike2Shp) and Grid files (Mike2Grd). These tools can be useful if you have manipulated your grid files in the MIKE Zero Grid Editor, since it does not directly support shape file export. Alternatively, you can open any dfs2 file in the MIKE SHE Setup Editor, as along as the unit type is the same) and then use the right mouse function to export to a shape file. If you want to convert a dfs3 file to a shape file or a grid file then you will need to extract a dfs2 file from the dfs3 first using the 2D Grid from 3D file tool that is found under the Extraction item in the MIKE Zero Toolbox.

Some items in the MIKE SHE Setup Editor do not support shape files. Mostly these are related to integer grid codes, such as Drain codes. In this case, it is difficult to assign integer values based on grid independent polygons. In a complex setup, it would be very difficult to control which cells are being assigned to which code when the polygons do not coincide with the cell boundaries. In some areas, the model results could be very sensitive to the code assigned.

## **19.2 Integer Grid Codes**

Integer Grid Codes are required when Real data varies in time or when model functions, such as soil profiles and paved areas, are assigned to particular zones. Integer Grid Codes are always integer values and do not vary with time.

For information on entering Integer Codes see the Integer Grid Codes section.



The following is an outline of the parameters that require Integer Grid Codes.

### **Model Domain**

Integer Grid Codes are used to define the inactive areas both inside and outside the model domain. Inactive areas outside of the model and the edge of the model are defined in the Model Domain and Grid section, while inactive, subsurface areas inside of the model are defined as Internal boundary conditions.

### **Component Calculations**

Integer Grid Codes are used to delineate such things as paved areas. In this case, the integer code acts like a flag and the calculations that are done are different depending on how the flag is set.

### **Model Properties**

Integer Grid Codes are used to delineate areas with similar properties. In this case, the integer value defines the zone to which the cell belongs. Thus, it defines which set of model properties is to be assigned to the particular cell.

For example, a model may be divided into a five zones each with a different soil profile for the unsaturated zone. In this case, the data tree will expand under the model property to include five separate sub-branches, where the soil profile can be defined.

### **Time Series**

Integer Grid Codes are used to define zones for which Real data varies in time. Thus, a time series for a parameter, such as precipitation rate, can be assigned to a model zone. Similarly to the Model Properties above, the model tree will expand under the parameter to include a separate sub-branch for each zone, where the time series file can be defined.

### **Time Varying Integers**


Grid Codes and Integer values do not normally vary with time. If such parameters do vary in time, then you must divide the simulation into time periods and run each time period as a separate simulation, starting each simulation from the end of the previous simulation using the Hot Start options (see Simulation Period).

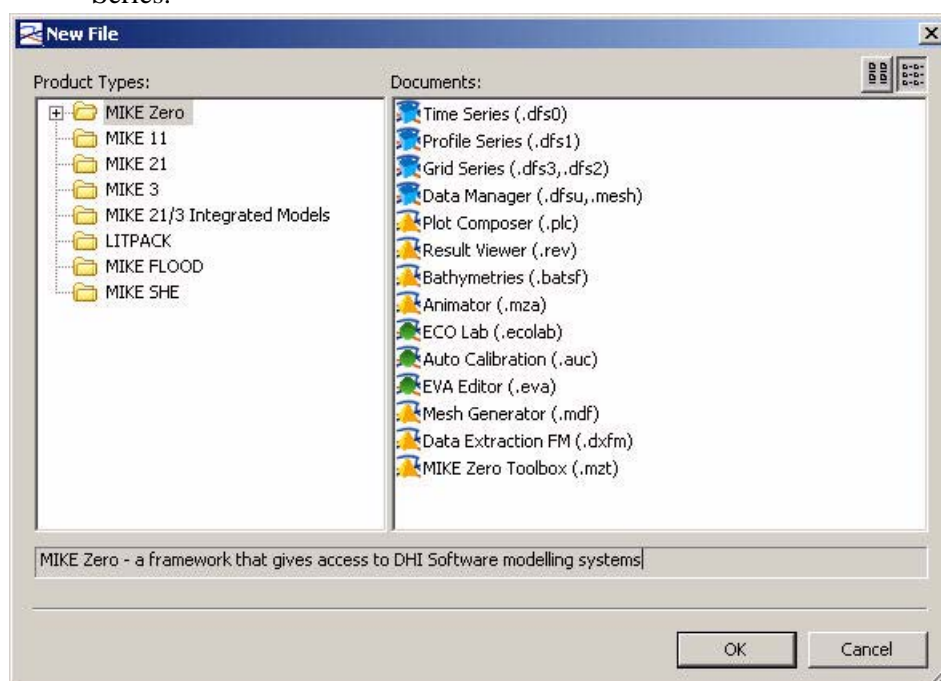


### 19.3 Gridded (.dfs2) Data

If the parameter is defined using gridded data, then the data must be in DHI's .dfs2 file format.

The easiest way to create the .dfs2 file is to use the **Create...** button, which creates a new grid with the proper default values and attribute type. You can then edit this grid in the MIKE Zero Grid Editor, which can be accessed using the **Edit...** button.

Alternatively, a .dfs2 file can be created using the Grid Series editor, which can be accessed by clicking on File|New in the pull-down menu, or using the New File icon, , in the toolbar, and then selecting Grid Series.



If you create the file from these tools you must be careful to ensure that the EUM Data Type matches the parameter that you are creating the file for. For more information on the EUM data types, see EUM Data Units.

The grid for the .dfs2 file does not have to be the same as the numerical model grid. However, if the grids are not subsets of one another then the grids will be interpolated using the bilinear interpolation during the pre-processing stage.



The parameter grid and the model grid are aligned with one another if the parameter grid or the model grid contain an even multiple of the other grid's cells. For example, if the parameter grid was two times finer, then every model grid cell must contain exactly four parameter grid cells.

If the grids are aligned then the parameter grid will be averaged to the model grid during the pre-processing stage. However, in some cases it does not make sense to average parameter values. For example, Van Genuchten soil parameters cannot really be averaged, since they are a characteristic of the soil. In such cases, you should ensure that the model grid and the parameter grid file are identical.

### 19.3.1 Stationary Real Data

Spatially distributed Real parameters, such as conductivity or topography, can be defined in three ways, namely they can be defined as a uniform (global) value or they may be distributed and defined using either gridded data (.dfs2 file), GIS points and polygons (ArcView .shp file), or irregularly distributed point data (x, y, value coordinate file).

#### Uniform

A uniform, global value means that all the grid cells in the model will have the same value.

#### GIS point and line data

If the parameter is defined using an ArcView shape (.shp) file, then the point and line data will be interpolated to the model grid during the pre-processing stage, using the interpolation method selected. The following interpolation methods are included:

- Bilinear Interpolation (*V.I p. 292*), or
- Triangular Interpolation (*V.I p. 296*).

It does not make sense to interpolate some parameters to the model grid. In such cases, the use of line and point data should be avoided.

#### Distributed Point data

If the parameter is defined using irregularly distributed point data, then the values at each point will be interpolated to the model grid during the pre-processing stage, using the interpolation method selected. The following interpolation methods are included:

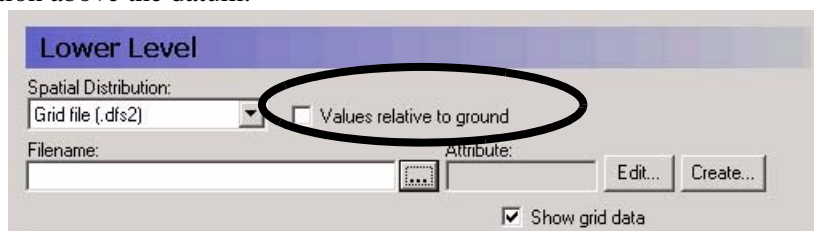
- Bilinear Interpolation, or
- Triangular Interpolation.



It does not make sense to interpolate some parameters to the model grid. In such cases, the use of line and point data should be avoided.

### Elevation Data

Elevation data, such as Layer elevations, is handled exactly the same as all other Stationary Real Parameters, except that the value may be optionally specified as a depth below the ground surface rather than absolute elevation above the datum.



**Note:** The value must be negative if it is below the ground level.

**Tip:** The current tools do not allow you to specify a polygon shape file with real values. However, this would be desirable in some cases, such as when implementing Mannings M values based on vegetation distributions. A trick to get around this limitation is the following:

- 1 Temporarily assign an integer grid code to each of the polygons.
- 2 Specify this file as an input file for one of the data items that needs integer grid codes, such as drain codes.
- 3 Right click on the map that will be displayed and save the map view to a dfs2 file
- 4 Open this dfs2 file in the grid editor and use the grid editor tools to replace the integer values with real values
- 5 In the Grid Editor, change the EUM unit to the appropriate value
- 6 Save the file and then load it into the Data item for which you wanted it.

### 19.3.2 Time-varying Real Data

If the time-varying Real parameter does not vary spatially then the parameter must be defined as Global with either a Fixed or Time-varying value (see Uniform + Constant and Uniform + Time Varying).

Often, time-varying data, such as precipitation rate, are spatially distributed using measurement stations, which in the model are translated into model zones using, for example, Thiessen polygons. In this case, each station is associated with a .dfs0 time series file that contains the time series





of precipitation rate. Station-based zones are defined using Integer Grid Codes in either a .dfs2 file as Grid Codes, or in a Shape (.shp) file as polygons with an Integer Code (see Station-based + Grid Codes or Polygons).

### Uniform + Constant

The parameter Value will be assigned to every cell in the model or layer as appropriate and will remain constant throughout the simulation.

### Uniform + Time Varying

The time series in the .dfs0 file will be assigned to every cell in the model or layer as appropriate.

### Station-based + Grid Codes or Polygons

Station-based time varying data means that the model domain is divided into zones that are defined by an Integer Grid Code.

If a .dfs2 file is used, then the Integer Grid Codes are defined on a regular grid, which is interpreted to the model grid during the Pre-processing stage.

If the Integer Grid Codes are defined using polygons then you must supply an ArcView .shp file containing polygons each with an Integer Grid Code. The item **Fill Gaps with:** allows you to define the Integer Grid Code to use in the event that a cell is not included within one of the polygons.

Once the file containing Integer Grid Codes has been defined, a new level in the data tree will appear below the current level, containing one entry for every unique Integer Grid Code in the file.

On this level, you must then supply a time series values for every Integer Grid Code. However, the time series can also be fixed, in the sense that a



constant value over time is used. This makes it easy to use detailed time series for some zones and constant values for zones where little information exists.

The time series dialogue itself includes two graphical views. The upper graphic displays the time series that is being applied and the lower graphic shows where the time series will be applied.

### **19.3.3 Integer Grid Codes**

The dialogues for Integer Codes function essentially same as those for Stationary Real Data, except that interpolation does not make sense for integer grid codes.

If Integer Grid Codes are being used to assign Model Properties, such as soil profiles or time series, then new sub-branches will appear in the data tree corresponding to the number of unique Integer Grid Codes in the .dfs2 file.

#### **Uniform Value**

A Uniform, global value means that all the grid cells in the model will have the same value. Thus, all cells would belong to the same zone.

#### **Grid File (.dfs2)**

If the Integer Code is defined using a grid file, then the Integer Code is defined on a grid. This grid may be different than the numerical model grid. However, the grids must be subsets of one another. That is, the Integer Code grid and the model grid must be aligned with one another and the Integer Code grid or the model grid must contain an even multiple of the other grid's cells. For example, if the Integer Code grid was two times finer, then every model grid cell must contain exactly four Integer Codes.

Normally, the Integer Code will be assigned to the model grid based on the most prevalent Integer Code in the cell. However, this can lead to problems when the a particular code is both infrequent and widely dispersed. For example, if a model area contained many small wetland areas that were much smaller than a grid cell.

For this reason, a bookkeeping count is kept of the assignments to reduce any bias in the assignment of Integer Codes and ensure that less frequently occurring Integer Codes will be represented in the resulting model grid. For example, if there were two different Integer Codes, A and B, used in the model and A always occurred more frequently in each model cell, the bookkeeping count would ensure that B would actually be assigned to some of the model cells. The final frequency of occurrence of the Integer



Codes in the model cells would reflect the underlying frequency of occurrence of the Integer Codes. That is, if A occurred twice as often as B, the model grid would also contain twice as many A's as B's.

Thus, in our widely dispersed wetland example, if every model grid cell contained 9 Integer Codes for Land Use, and 1/9 of the Land Use grid codes were for wetlands, then every ninth Model Cell would be assigned a Land Use grid code for wetlands.

### Polygons

In the current version, only some of the parameters are set up to accept .shp file polygons. Currently, .shp file polygons are only allowed in:

- Model Domain and Grid (V.2 p. 52)
- Precipitation Rate (V.2 p. 58)
- Vegetation (V.2 p. 64),
- Reference Evapotranspiration (V.2 p. 79)
- UZ Soil Profile Definitions (V.2 p. 93),
- SZ Internal boundary conditions (V.2 p. 121), and
- Horizontal Extent (V.2 p. 111) of SZ Lenses.

**Note** The Horizontal Extent (V.2 p. 111) of SZ Lenses accepts polygons, but the dialogue is still set up for point/line .shp files and an error is given in the Data Verification window.

Model grid codes are assigned based in which polygon the centre of the cell is located in.

## 19.4 Interpolation Methods

The gap filling is based on the concept that we have to calculate the depth in the point  $(x_c, y_c)$ . We define this as the function  $z_c = f(x_c, y_c)$ . If we place our self in this point, we can divide the world up into four quadrants Q1 - Q4. From here it's a matter of finding some points from the raw data set relatively close to this point. The search radius for all possible techniques can be entered - in grid cell distance. Points outside this distance will never be taken into account.

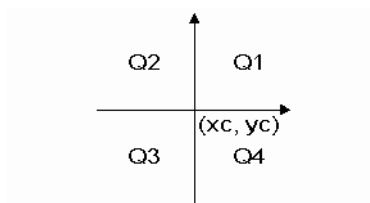


Figure 19.1 Definition of quadrants

### 19.4.1 Bilinear Interpolation

This technique finds four points from the raw data set - one in each quadrant. The search is done in the following way. A mask of relative indices is created. The cells in this mask are sorted according to the distance. For the quadrant Q1 the cells are sorted in the following way, the grid point it self being excluded.

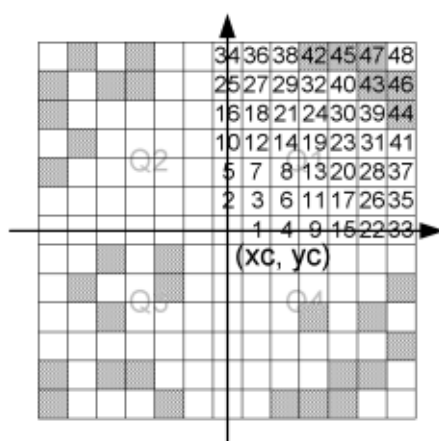


Figure 19.2 Illustration of the neighbouring grid cells being sorted

Note that the grid cells with a crosshatch pattern contain raw data points. When the closest raw data point in each quadrant is found, we have four points that form a quadrangle. This quadrangle contains the centre point, where we want to calculate the z-value. This is illustrated on Figure 19.3.

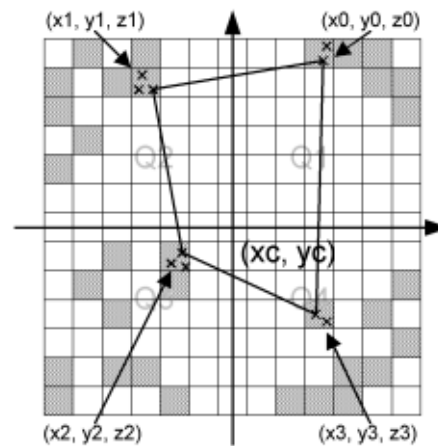


Figure 19.3 Illustration of the closest raw data points in each quadrant.

Note that each grid cell might contain more raw data points. If this is the case, the closest of these is chosen. We now have an irregular quadrangle, where the elevation is defined in each vertex. We need to compute the elevation in  $(x_c, y_c)$ . If we transform our quadrangle into a square, we can perform bilinear interpolation. This is illustrated on Figure 19.4.

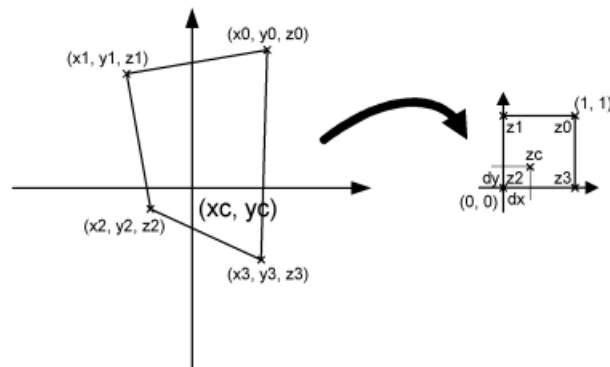


Figure 19.4 Illustration of bilinear interpolation.



First the interpolation requires the transformation from quadrangle to a normalized square. This is done in the by computing 8 coefficients in the following way:

$$\begin{aligned}A_1 &= x_0 \\A_2 &= y_0 \\B_1 &= x_1 - x_0 \\B_2 &= y_1 - y_0 \\C_1 &= x_3 - x_0 \\C_2 &= y_3 - y_0 \\D_1 &= x_2 - x_1 + x_0 - x_3 \\D_2 &= y_2 - y_1 + y_0 - y_3\end{aligned}\tag{19.1}$$

Mapping the coordinates  $(x_c, y_c)$  to the normalized square  $(dx, dy)$  is done by solving equation (19.2).

$$ax^2 + bx + c = 0\tag{19.2}$$

where the coefficients are

$$\begin{aligned}a &= D_1B_2 - D_2B_1 \\b &= D_2x_c - D_1y_c - D_2A_1 + D_1A_2 + C_1B_2 - C_2B_1 \\b &= C_2x_c - C_1y_c + C_1A_2 - C_2A_1\end{aligned}\tag{19.3}$$

Solving equation (19.2) gives us  $dx$ .

$$dx = \frac{-b \pm \sqrt{b^2 - 4ac}}{2a}\tag{19.4}$$



where  $0 \leq dx \leq 1$  is used to choose the correct root.  $dy$  can now be computed in two ways:

$$dy = \frac{x_c - A_1 - B_1 dx}{C_1 - D_1 dx} \quad (19.5)$$

or

$$dy = \frac{x_c - A_2 - B_2 dx}{C_2 - D_2 dx} \quad (19.6)$$

Choosing between (19.5) and (19.6) is done in such a way, that division by zero is avoided.  $(x_c, y_c)$  has been mapped to  $(dx, dy)$ . The task was to compute the elevation in the point  $(x_c, y_c)$  and this is done in the following way using regular bilinear interpolation:

$$z_c = (1 - dx)(1 - dy) z_2 + dx(1 - dy) z_3 + (1 - dx)dy z_1 + dxdy z_0 \quad (19.7)$$

If less than four points are found (if one or more quadrants are empty), the double linear interpolation is replaced with reverse distance interpolation (RDI). This is done according to the following scheme:

$$w_i = \frac{1}{\sqrt{(x_i - x_c)^2 + (y_i - y_c)^2}} \quad (19.8)$$

$$w_s = \sum_{i=1}^N w_i \quad (19.9)$$

$$z_c = \frac{1}{w_s} \sum_{i=1}^N w_i z_i \quad (19.10)$$

The method works fairly efficiently, but it has one drawback. The quadrant search is heavily dependent on the orientation of the bathymetry. If the bathymetry is rotated 45 degrees 4 completely different points might be used for the interpolation. For this reason there is also a Triangular

interpolation method, which can be used, and this method should be direction independent.

### 19.4.2 Triangular Interpolation

As mentioned previously the ‘Bilinear Interpolation’ is dependent on the orientation of the bathymetry. The ‘Triangular Interpolation’ is made as an answer to this problem. First the closest point to  $(x_c, y_c)$  is found. The following figure shows this:

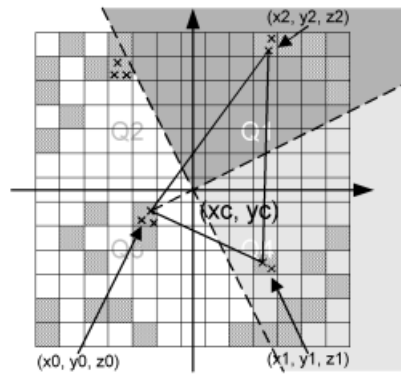


Figure 19.5 Illustration of triangular interpolation

In this example the point  $(x_0, y_0, z_0)$  is the closest point. When this point is identified, two quadrants are identified – indicated by the light grey and the dark grey areas. The closest point in these two quadrants are then found. They can be seen on the figure as  $(x_1, y_1, z_1)$  and  $(x_2, y_2, z_2)$ . The interpolation is then done in two steps. First the coefficients describing the plane defined by the 3 found points are computed:

$$\begin{aligned}
 A &= \frac{-(y_1 - y_0)(z_2 - z_0) + (y_2 - y_0)(z_1 - z_0)}{(x_1 - x_0)(y_2 - y_0) - (x_2 - x_0)(y_1 - y_0)} \\
 B &= \frac{(x_1 - x_0)(z_2 - z_0) + (x_2 - x_0)(z_1 - z_0)}{(x_1 - x_0)(y_2 - y_0) - (x_2 - x_0)(y_1 - y_0)} \\
 C &= z_0 - Ax_0 - By_0
 \end{aligned}
 \tag{19.11}$$

And secondly the actual interpolation is done:

$$z_c = Ax_c + By_c + C
 \tag{19.12}$$





If less than 3 points are found, reverse distance interpolation (RDI) is used. The triangular interpolation is more time consuming due to the more complex direction independent search, but better end results should be achieved with this method.

## 19.5 Node numbering in the Grid Editor

The Grid Editor is a generic grid tool for all DHI Software, and was originally developed for the Marine programs MIKE 21 and MIKE 3. However, this often leads to confusion in the node and layer numbering because MIKE 21 and MIKE 3 use a different numbering system.

### Node numbering in the Grid Editor

In the Grid Editor (and in MIKE 21 and MIKE 3) the nodes are numbered starting in the lower left from (0,0), whereas in MIKE SHE the nodes are numbered starting in the lower left from (1,1).

### Layer numbering in the Grid Editor

In the Grid Editor (and in MIKE 21 and MIKE 3) the layers are numbered starting at the bottom from 0, whereas in MIKE SHE the layers are numbered starting at the top from 1.

## 19.6 Performing simple math on multiple grids

In the upper menu of the Grid Editor, under tools, there is an item called **Copy File into Data**.





If you select this item then a dialogue appears where you can insert an existing dfs2 or dfs3 file into the current dfs2 or dfs3 file that you are editing in the Grid editor.

|   | Source Item | Maps to                             | Target Item |
|---|-------------|-------------------------------------|-------------|
| 1 | Lower Level | <input checked="" type="checkbox"/> | Topography  |

Alternatively, you can define an operation that you want to do with the file. For example, if you were editing a topography file, you could subtract all of the values in a lower elevation file, to obtain a thickness distribution for a layer.

The principle advantage of this tool, is that time varying dfs2 and dfs3 files can be manipulated. However, if the operations are complex, but not time varying then

### Target file

The target file is the current file you are editing in the Grid editor. The operations that you do are performed on the target file. So, if you don't want to edit the target file, copy it to a new name first and edit the copy.

### File to Copy

The top section of the dialogue is the name of the source file that you want to insert into, subtract from, add to, etc. the target file.

### Item mapping

If the target file or the source file has more than one item in it, then all of the items will be listed here and you will be able to choose whether or not to map the various items to one another.



## 2D to 3D Layer Mapping

If you are mapping a 2D dfs2 file into a 3D dfs3 file, then you can choose to map all of the layers or only a single layer.

### Sub-area position

You select to map the source file onto the target file starting at a different location than the origin. In this case, you must specify the coordinates in the target grid where the origin of the source grid should be positioned. For example, if you have a 20x20 grid and we wish to copy data into the 4x4 rectangle given by the four nodes (10,14), (13,14), (13,17) and (10,17), then you should select a 4x4 grid file and specify j-origin=10 and k-origin=17. **Note: the Grid editor starts its nodal numbering at 0,0.**

### Time Position

The source grid and target grid do not have to have equal time steps or the same time origin. In this section of the dialogue, you can specify the time at which the source grid should be added to the target grid. In this way, you can add additional time steps to the end of a time varying dfs2 file, or insert hourly information into a monthly time series, for example.

### Operation

Finally, you can specify how the source grid file should interact with the target file.

**Copy** - all values are copied such that they replace the existing data in the data set

**Copy if target differs from delete value** - values in the source file will be copied into the target file, only if the target value is a delete value

**Copy if source differs from delete value** - values in the source file will be copied into the target file, only if both the source value and the target values are not delete values

**Copy if source AND target differs from delete value** - values in the source file will be copied into the target file, only if the source value is not a delete value

**+** - the source values will be added to the target values

**-** - the source values will be subtracted from the target values

**\*** - the source values will be multiplied by the target values



/ - the source values will be divided by the target values

## **19.7 Performing complex operations on multiple grids**

In the MIKE SHE Toolbox (V.2 p. 203), under Util, there is a Grid calculator (V.2 p. 205) tool, which allows you to perform complex operations on .dfs2 grid files. However, the grid files must have the same grid dimensions and they may not include multiple time steps or multiple items. Thus, this tool is much more restrictive than the grid operations available in the Grid Editor. However, you can make complex chains of operations and save the setup, which can save you a lot of time if you are doing the same operation many times or after each simulation.

The Grid Calculator works like a wizard, with Next and Back buttons to move between dialogues.



*Phi Software*

**GRID EDITOR**





## 20 INTRODUCTION

The appearance of the Grid Series Editor differs if you create a new (blank) time series compared to opening an existing data (\*.dfs2) file.

Creating a new grid (or matrix) series requires specification of properties for the grid series file, and the File Properties dialog is therefore opened in this case.

If you are opening an existing grid series data file, the data are immediately presented in the Grid Series data dialog where data can be viewed and edited both in a graphical and in a tabular view. You cannot directly change the already defined file properties of a grid series.

You operate the Grid Series Editor from the main menu, the tool bar icons, or by right-clicking on the graphical view.

### 20.1 Create a New Dataset

To create a new dataset containing a 2D or a 3D grid, go to File New and select Grid Series under the MIKE Zero heading. This will open the New Grid Dialog shown below.

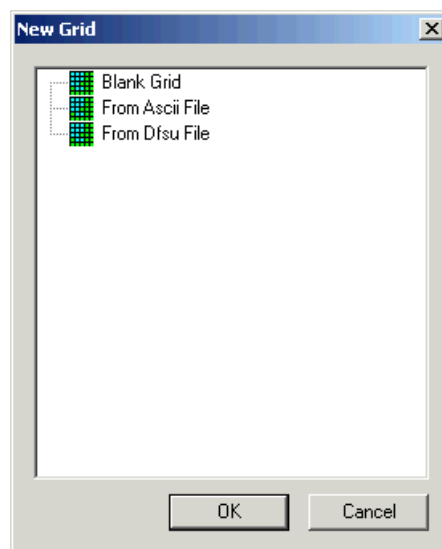


Figure 20.1 New Grid dialog



By selecting *Blank Grid* you must define the grid parameters as described in the section *New File* (p. 309).

By selecting *From ASCII File* you must follow the instructions given in section *Import from ASCII File* (p. 311).

By selecting *From Dfsu File* you must follow the instructions described in the section *Import from Dfsu File* (p. 312).

## 20.2 Open an Existing Dataset

To open an existing dataset, go to File Open and select the file format that you are looking for. If you double-click a file in the Windows Explorer with a file format associated with the Grid Editor, then the editor will open and load the data ready for editing.

## 20.3 Editing the Dataset

When you have selected the dataset to edit, the editor is ready to work. Similar to other MIKE Zero DFS editors, such as the Time Series Editor and the Profile Editor, the Grid Editor has two views, a graphical view and a tabular view. A movable splitter bar allowing you to adjust the relative sizes of the two views separates these views.

## 20.4 Further help

Further help can be found for the following topics:

- Graphical View (p. 322) - about the graphical view of the data
- Tabular View (p. 321) - about the tabular view of the data
- New File (p. 309) dialog - to create a new dataset

### Tools

- Navigation (p. 331) - to navigate in the dataset in time and how to select an item
- Set Value (p. 335) - to assign a value to a selection of grid cells
- Filter (p. 335) - to filter a selection of data (e.g. smooth the data)
- Crop (p. 341) - to discard some of the data and keep the rest





- Calculator (*p. 336*) - transform the data using expressions that involve the data itself, the time step etc.
- Interpolation (*p. 334*) - fill blank values
- Copy File into Data (*p. 340*) - import data from another dataset
- Toolbars (*p. 328*) - the Windows toolbars associated with the Grid Editor





## 21 FILE

### 21.1 File Formats

The Grid Editor supports two file formats, the DFS format and ASCII.

Note that files can be converted from one format to the other, i.e. saved in a different format than they opened in, but with restrictions.

#### 21.1.1 DFS format

This format is developed by DHI for storage of hydrodynamic data saved in grids. Both one, two and three dimensional data can be stored, although only two and three dimensional data are relevant for the grid editor.

Files saved in this format must have the extension \*.dfs2, \*.dfs3, \*.dt2 and \*.dt3.

#### 21.1.2 ASCII format

This is a general format which can be generated by most spread sheets or text editors. The first part of an ASCII File must contain the Header Information. An example of a Header Information is given below:

```
Title Example of Import from ASCII File
Dim 3
Geo UTM -1.20 37.32 -35.40
Time EquidistantTimeAxis 2000-05-03 00:00:00 2 30.00
NoGridPoints 6 4 5
Spacing 200.00 200.00 1
NoStaticItems 1
Item Bathymetry Undefined Undefined
NoDynamicItems 2
Item H (m) waterdepth Water level meter
Item Q (m**3/s/m) flux Flow Flux m^3/s/m
Delete -1E-030
<Empty line>
```

The lines in the Header Information must appear in this order and fields must be separated with tabs. The first item in the lines of the Header Information is a string, i.e. 'Title', 'Dim', 'Geo' etc. The Header Information ends with an empty line.

The lines specify the following:



Title

Dimension: Can be either 2 or 3.

Geographic Information: UTM-zone, origin longitude, origin latitude and orientation.

Time Information: A Time Axis Description, Start Date yyyy-mm-dd, Start Time, number of Time Steps and Time Interval in seconds.

Grid Information: Number of Grid points in first, second and third dimension.

Number of Static Items (a Static Item is data related to time series data, but not a part of it. For example, the bathymetry in a file from a HD-run with H, P Q as Dynamic Items).

Static Items: Name, Type, Unit (these can be left unspecified).

Number of Dynamic Items.

Dynamic Item: Name, Type, Unit (these can be left unspecified)

Delete value: The Delete value should be a number not typical of the data

Data follows after the Header Information and must be arranged in blocks. An example of a data block is shown below:

```
tstep 187 item 1 layer 0
20 20.0313 21.093 21.1076 21.2892 -1E-030
20 20.2856 21.1294 21.1587 21.2748 -1E-030
-1E-030 -1E-030 -1E-030 21.0826 21.2092 -1E-030
-1E-030 -1E-030 -1E-030 20.8634 20.8749 -1E-030
<Empty line>
```

The first data blocks must contain the Static Items. The Dynamic Items follow and must be presented in the following order:

```
tstep 0 item 1 layer 0
tstep 0 item 1 layer 1 - until all layers are specified
tstep 0 item 2 layer 0
tstep 0 item 2 layer 1 - until all Dynamic Items are specified
tstep 1 item 1 layer 0
Etc.
```

Files saved in this format can have any extension although dfs2, dfs3, dfs, dt2 and dt3 should be avoided.

### 21.1.3 Grid State Format

You can save your grid file as a Grid State File (.gsf). This will save your normal grid file together with extra information about a specific palette and background layers if any.

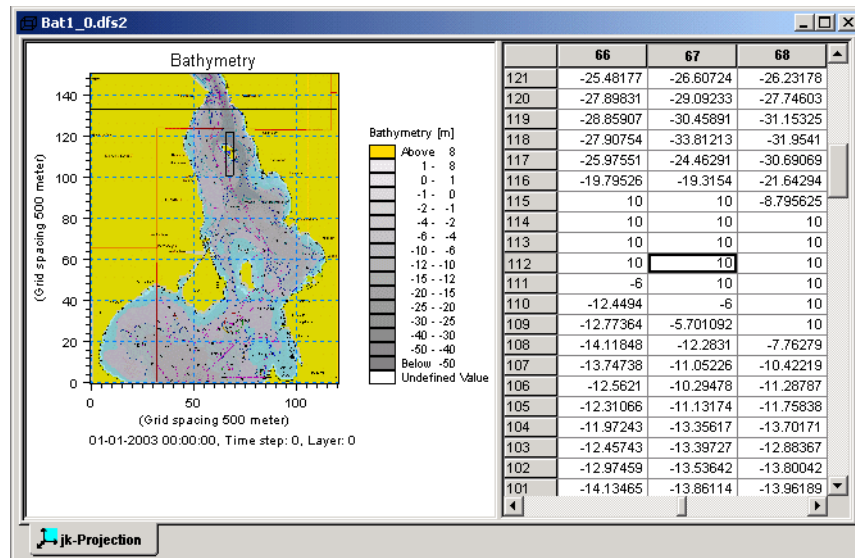


Figure 21.1 Example of saved grid file in gsf-format as interpreted by the grid editor.

## 21.2 New File

The new-dialog is used to create new 2D and 3D grids. A wizard takes you through five steps to input the basic information needed to define the grid.

### 21.2.1 Step 1: Select the type of grid to be created

2D and 3D refer to the number of dimensions of the grid in space. Both types of grids can have one or more time steps, see step 3.

### 21.2.2 Step 2: Specify the projection, the geographical position of the origin of the dataset and the orientation

Choose the projection system you will use for the grid. If UTM is selected, choose the zone number. The UTM zone number is defined by:

$$\text{UTM Zone} = \text{integer part of} \left( \frac{180 + \lambda}{6} + 1 \right)$$

where  $\lambda$  is the longitude.

There are three options for specifying the geographical position of the origin of the new grid:

- specify in latitude and longitude and input in degrees, minutes and seconds
- specify in latitude and longitude and input in decimal degrees



- specify in Universal Transverse Mercator (UTM) coordinates by giving the UTM system and the Easting and Northing of the origin in meters. When you select a general UTM zone, the zone number is required. When using local zones as for example Hong Kong Grid (HKG), then the zone number is not required.

The geographical position of the origin of the grid is defined as the centre point in the first grid cell  $(j,k) = (0,0)$ .

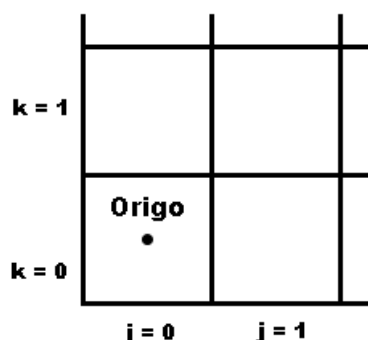


Figure 21.2 Definition of Origo

Furthermore, the orientation of the grid at the origin must be specified. This is defined as the angle between true north and the y-axis of the grid measured clockwise. A mnemonic way of remembering this definition is by thinking of NYC, which normally means New York City, but which for our purpose means "from North to the Y-axis Clockwise", see the figure below.

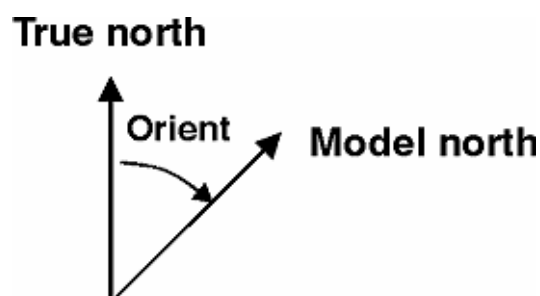


Figure 21.3 Definition of grid orientation



### 21.2.3 Step 3: Specify the temporal and spatial properties

To define the temporal properties you need to specify:

- the type of axis, only choice is equidistant, meaning a constant time step between the grids
- the time for the first grid
- the time step between the grids, given in seconds
- the number of time steps in the dataset

The spatial properties include:

- the number of grid points in each direction
- the grid spacing in meters in each direction. The grid spacing is constant in each direction, but can vary between directions

### 21.2.4 Step 4: Specify the items to be included

The grid can contain several items. Give the item(s) a name and select the item type from the list. A unit has been defined for each item type, and this unit will automatically be set when choosing the item type. If you cannot find the item type that you need, then use 'undefined'. Using the 'Insert', 'Append' and 'Delete' options you can manage the list of items.

All values in the grid will be initially set to the delete value specified on this page. This means that values that are not somehow filled later in the editing process will have the delete value.

### 21.2.5 Step 5: Overview

Press 'Finish' to complete the initialisation process and start editing.

## 21.3 Import from ASCII File

This tool can be used to import grid data from an ASCII File and convert it into a DFS File (\*.dfs2, \*.dfs3, \*.dt2 and \*.dt3). The DFS File format is developed by DHI for storage of hydrodynamic data.

These files typically contain model grids, wind fields, model results etc.

To import grid data, go to 'File', 'New' and select 'Grid Series' under the MIKE Zero heading. This will open the 'Grid Series' dialog. Choose 'From ASCII File', and press 'OK'.



### **21.3.1 File to Import**

On the 'Import From ASCII' dialog select the ASCII File from which you wish to import the data. The ASCII File must have a certain format in order to be read correctly.

### **21.3.2 Completion and Editing**

To complete the importing of data press 'OK'. Thereafter the new DFS File is shown with the Grid Editor. Here you can edit data and finally save the DFS File.

### **21.3.3 Hint**

Creating a frame from an existing file can make it easier to create the header for the ASCII file to be imported. Simply by exporting an existing file to ASCII and change the relevant header information and include the actual data after the header file.

## **21.4 Export to ASCII File**

You can Export a Grid File in DFS Format to an ASCII File. For further description of Grid File Formats, see File Formats (*p. 307*).

Open the DFS File (\*.dfs2, \*.dfs3 or \*.dfs) in the Grid Editor. Go to 'File' and 'Export to ASCII'. In the pop-up window you can specify where the ASCII File should be saved and under which File Name.

## **21.5 Import from Dfsu File**

This tool can be used to import grid data from a Dfsu File and convert it into a DFS File (\*.dfs2, \*.dfs3, \*.dt2 and \*.dt3).

### **21.5.1 Step 1: Select File to Import**

On the 'Import from dfsu file' dialog select the Dfsu File from which you wish to import the data.

### **21.5.2 Step 2: Specify geographical parameters**

You must specify the geographical position of the origin of the new grid and the orientation of the grid.

The projection of the geographical coordinate system is read from the Dfsu file.

See Geographical Information (*p. 315*) for further information.





### **21.5.3 Step 3: Specify the spatial properties**

You must specify the number of grid points and the grid spacing in each direction, respectively.

The time axis properties is read from the Dfsu file.

### **21.5.4 Step 4: Specify land value**

You must specify the minimum value that defines land.

The items are read from the Dfsu file.

### **21.5.5 Step 5: Completion and Editing**

To complete the importing of data press 'Finish'. Thereafter the new DFS File is shown with the Grid Editor. Here you can edit data and finally save the DFS File.





## 22 EDIT

### 22.1 Geographical Information

The Geographical Information dialog is used to set the geographical position and orientation of the grid, as well as the projection zone.

**Edit Properties**

**Geographical Information**

Map Projection  
Type: UTM-33

Geographical Position of Origin

☒ Deg., Min., Sec. ☐ Decimal Deg. ☐ Projection

Deg., Min., Sec.

Longitude: 12 [deg] 26 [min] 19.4697620156 [sec] Latitude: 55 [deg] 13 [min] 32.5482327256 [sec]

Decimal Deg.

Longitude: 12.4387416006 [deg] Latitude: 55.2257078426 [deg]

Projection

Type: UTM-33

Easting: 337100.000026 [m] Northing: 6122900.00176 [m]

Orientation at Origin

Orientation: 327 [deg]

OK Cancel Help

There are three options for specifying the geographical position of the origin of the grid:

- specify in latitude and longitude and input in degrees, minutes and seconds
- specify in latitude and longitude and input in decimal degrees



- specify in Universal Transverse Mercator (UTM) coordinates by giving the UTM system and the Easting and Northing of the origin in meters. When you select a general UTM zone, the zone number is required. When using local zones as for example Hong Kong Grid (HKG), then the zone number is not required.

The geographical position of the origin of the grid is defined as the center point in the first grid cell  $(j,k) = (0,0)$ .

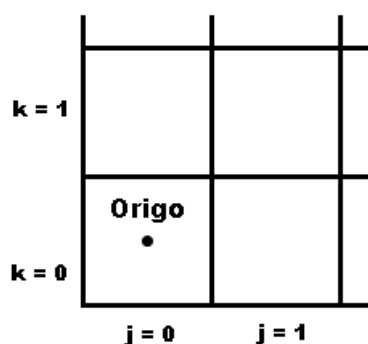


Figure 22.1 Definition of Origo

Furthermore, the orientation of the grid at the origin must be specified. This is defined as the angle between true north and the y-axis of the grid measured clockwise. A mnemonic way of remembering this definition is by thinking of NYC, which normally means New York City, but which for our purpose means "from North to the Y-axis Clockwise", see the figure below.

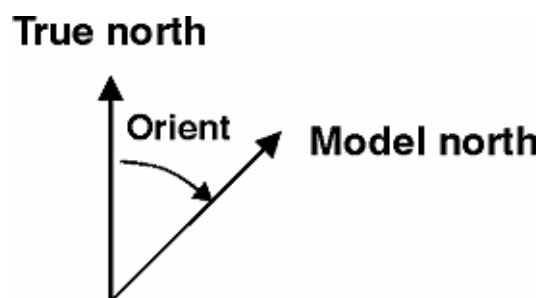
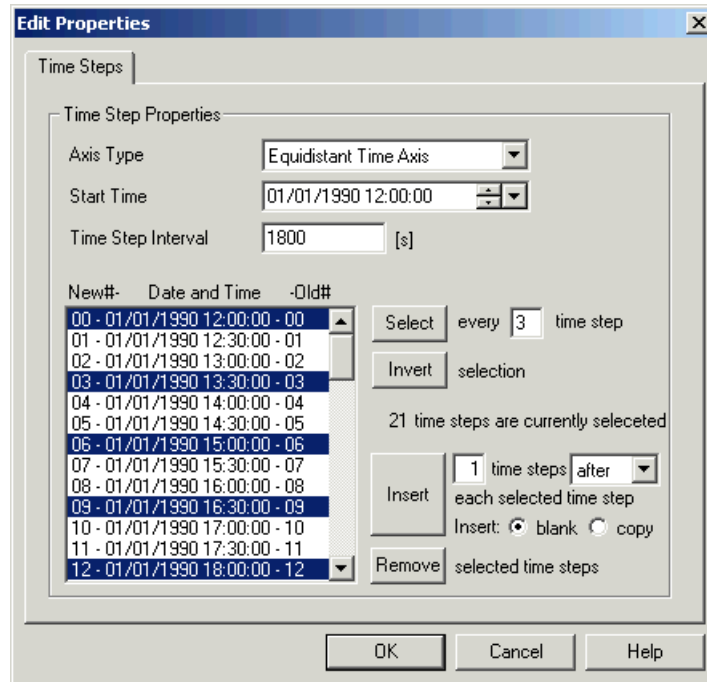


Figure 22.2 Definition of grid orientation



## 22.2 Time Steps

The Time Steps dialog is used to manage the time axis of the file.



It is possible to specify the start date and time for the file as well as the time step interval between the time steps.

### 22.2.1 Modifying Time Steps

#### Selecting time steps

It is possible to select time steps in the dialog by simply clicking on them in the list, or by using the "Select" and/or "Invert" button. The number of selected time steps are displayed next to the list.

#### Inserting time steps

- 1 Select the items in the list where the new items should be placed before/after
- 2 Specify the number of items to insert before/after the selected time steps
- 3 Specify whether the new items should be inserted before or after the selected items



- Specify whether the new items should be filled with blanks (delete values) or a copy of the values from the selected items they are placed before/after
- Press the "Insert" button.

### Removing time steps

To remove one or more time steps simply select them as described above and press the "Remove" button.

## 22.3 Items

The Items dialog is used to manage the items in the file.

**Edit Properties**

Items

Item Information

|   | Name                      | Type        | Unit                |
|---|---------------------------|-------------|---------------------|
| 1 | H Water Dept              | Water Level | meter               |
| 2 | P Flux m <sup>3</sup> /s/ | Flow Flux   | m <sup>3</sup> /s/m |
| 3 | Q Flux m <sup>3</sup> /s/ | Flow Flux   | m <sup>3</sup> /s/m |

Insert Append Delete Item Filtering...

Delete value:

-0.001

OK Cancel Help

The grid can contain several items. Give the item(s) a name and select the item type from the list. A unit has been defined for each item type, and this unit will automatically be set when choosing the item type. If you cannot find the item type that you need, then use 'undefined'. Using the 'Insert', 'Append' and 'Delete' options you can manage the list of items.

For new items all values in the grid will initially be set to the delete value specified on this page. This means that values that are not somehow filled later in the editing process will have the delete value.



### 22.3.1 Editing an existing file

#### Copy

- 1 Insert or append a new item.
- 2 Select the item (row) to copy by clicking on the left most (grey) cell and press Ctrl+C.
- 3 Select the newly created item (row) by clicking on the left most (grey) cell and press Ctrl+V.

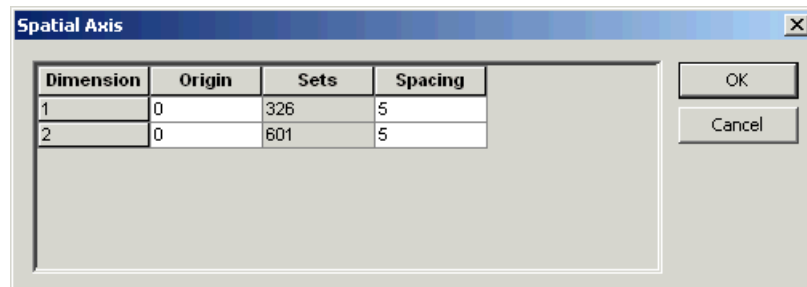
The properties and values of the new item will be identical to the original item

#### Delete value

It is not recommended to change the delete value in an existing file since all un-specified values are set to the delete value. After a change, these values will be interpreted as non-delete value.

## 22.4 Spatial Axis

The Spatial Axis dialog is used to manage the spatial axis of the file.



Each axis is defined by origin grid number, number of grid points and grid spacing. It is not possible to edit the number of grid points for an existing grid series file.

## 22.5 Edit Custom Blocks

This tool can be used to edit the custom blocks of a file. Custom blocks are data in dfs files where miscellaneous information about the file is kept.



### 22.5.1 MIKE 21 / MIKE 3 specific

Most MIKE 21 and MIKE 3 data files have a custom block called "M21\_misc". It is a block of 7 floats

- Item 1: orientation at origin relative to true north.
- Item 2: drying depth.
- Item 3: code for identifying whether or not the data contains geographical information; it is -900 if it contains geographical information.
- Item 4: the zland value, the value above which bathymetric data (the data itself in case of bathymetry data file; the prefix record containing the bathymetry in other cases) is considered as land.
- Item 5-6: are more free and may have different meaning in different situations.

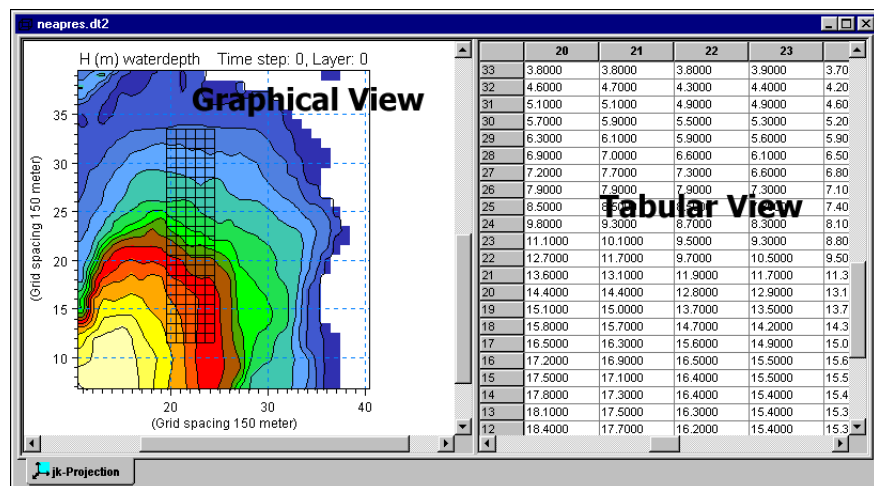




## 23 VIEW

The graphical view appears to the left-hand side of the Grid Editor editor. On the right-hand-side you have the Tabular View.

You may adjust the relative size of the graphical view and the tabular view by adjusting the splitter bar between the views.



### 23.1 Tabular View

The area that is viewed in the tabular view can be indicated in the Graphical View by switching on the selection overlay.

The tabular view works like a simple spreadsheet:

- Click on a cell, and you can enter or modify a value
- Use the mouse or the arrows to highlight a block of cells. The use cut (ctrl X), copy (ctrl C) and paste (ctrl V) as usual. These actions may be reversed by using undo (ctrl Z)
- You can move around in the table by using the arrow buttons or the TAB or ENTER keys. If the TAB and ENTER key is pressed at the right most column, the active cell is moved to the first column in the next line. This can be used to quickly enter data in a typewriter fashion. SHIFT+TAB or ENTER works the other way



### Cell Format

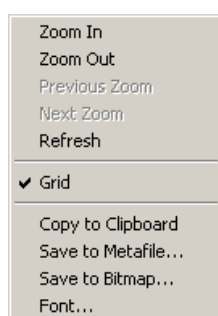
You can define the cell format by selecting **Tabular View** at the bottom of the drop-down menu for **View**.

### Decimal point

Please note that the tabular view always represents numbers with a '.' as the decimal point, regardless of which representation is used by the Windows system. You should be aware of this when copying numbers to and from other applications that may use a different notation.

## 23.2 Graphical View

In the graphical view you may manipulate the data using various tools. Inside the graphical view you can zoom and pan. There are three ways of zooming in and out: from the menu item *View*, from the standard toolbar, or clicking the right mouse button while pointing at the graphical view. The last action will bring up the following pop-up menu.



In 3D you may select the jk-plane, the jl-plane or the kl-plane by selecting the appropriate property page at the bottom of the graphical view.

### Graphical view settings

There are a number of graphical view attributes that can be set. These are accessible from the menu item *View*.

## 23.3 Palette

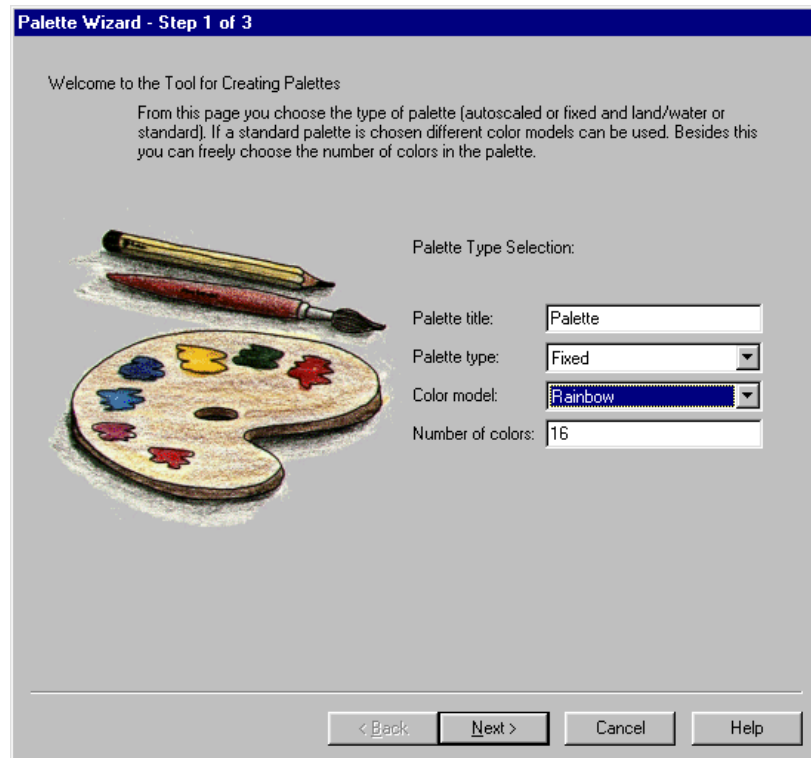
The color palette is used to specify the colours used for shading of the graphical view according to contour levels. A palette may be created and saved for later use.



If you want to save the palette together with the specific grid file, you can use the Grid State Format (*p. 308*).

A new palette is created using the Palette Wizard as described below.

### 23.3.1 Palette Wizard, step 1 of 3



#### Palette title

Give the palette a title.

#### Palette type

Choose the type of palette among two general type palettes:

- Linear auto scaled - a linear scale between the minimum and the maximum value in the view, automatically scaled
- Fixed - a scale where you can choose the intervals freely and where the scale does not change according to the data shown in the view

and two palettes which are suitable for plotting bathymetries:



- Land/water auto scaled - as the linear auto scaled with the extension that a special colour is specified for land areas
- Land/water fixed - as the fixed type with the extension that a special colour is specified for land areas

### Colour model

A number of different colour models are provided for convenience. These colours may then be modified during the second step. When using the land/water palette types, the colour model is fixed and cannot be changed.

### Number of colours

Choose the number of colours, i.e. the number of intervals in the palette.

## 23.3.2 Palette Wizard, step 2 of 3

**Palette Wizard - Step 2 of 3**

Please Specify the Color Table in the Palette

From this page you specify actual color table in the palette. The easiest way to choose the color is to check the actual color in the second column and the press the "..."-button and then choose from the color-dialog. For a fixed palette the corresponding value can be entered too. Note that the value defines the upper boundary of the interval.

Palette Table:

|    |                                     | Color | Red | Green | Blue | Value   |
|----|-------------------------------------|-------|-----|-------|------|---------|
| 1  | <input checked="" type="checkbox"/> |       | 255 | 0     | 0    | -1e-035 |
| 2  | <input type="checkbox"/>            |       | 0   | 0     | 0    | -1e-035 |
| 3  | <input type="checkbox"/>            |       | 0   | 0     | 0    | -1e-035 |
| 4  | <input checked="" type="checkbox"/> |       | 255 | 255   | 0    | -1e-035 |
| 5  | <input type="checkbox"/>            |       | 0   | 0     | 0    | -1e-035 |
| 6  | <input type="checkbox"/>            |       | 0   | 0     | 0    | -1e-035 |
| 7  | <input checked="" type="checkbox"/> |       | 0   | 255   | 0    | -1e-035 |
| 8  | <input type="checkbox"/>            |       | 0   | 0     | 0    | -1e-035 |
| 9  | <input type="checkbox"/>            |       | 0   | 0     | 0    | -1e-035 |
| 10 | <input checked="" type="checkbox"/> |       | 0   | 166   | 166  | -1e-035 |
| 11 | <input type="checkbox"/>            |       | 0   | 0     | 0    | -1e-035 |
| 12 | <input type="checkbox"/>            |       | 0   | 0     | 0    | -1e-035 |
| 13 | <input checked="" type="checkbox"/> |       | 0   | 0     | 255  | -1e-035 |
| 14 | <input type="checkbox"/>            |       | 0   | 0     | 0    | -1e-035 |
| 15 | <input type="checkbox"/>            |       | 0   | 0     | 0    | -1e-035 |
| 16 | <input checked="" type="checkbox"/> |       | 128 | 0     | 128  | -1e-035 |

Land Color:

Land Value:

< Back   Next >   Cancel   Help

### Palette Table

In the example above we have chosen 16 colours and used *Rainbow* as the colour model in step 1. We can now select any number of these levels and modify the colour and enter the value we want for this level. Note that the value defines the upper boundary of the interval. The colour can be speci-



fied either by the RGB-value or by opening a colour dialog by pressing the "..."-button.

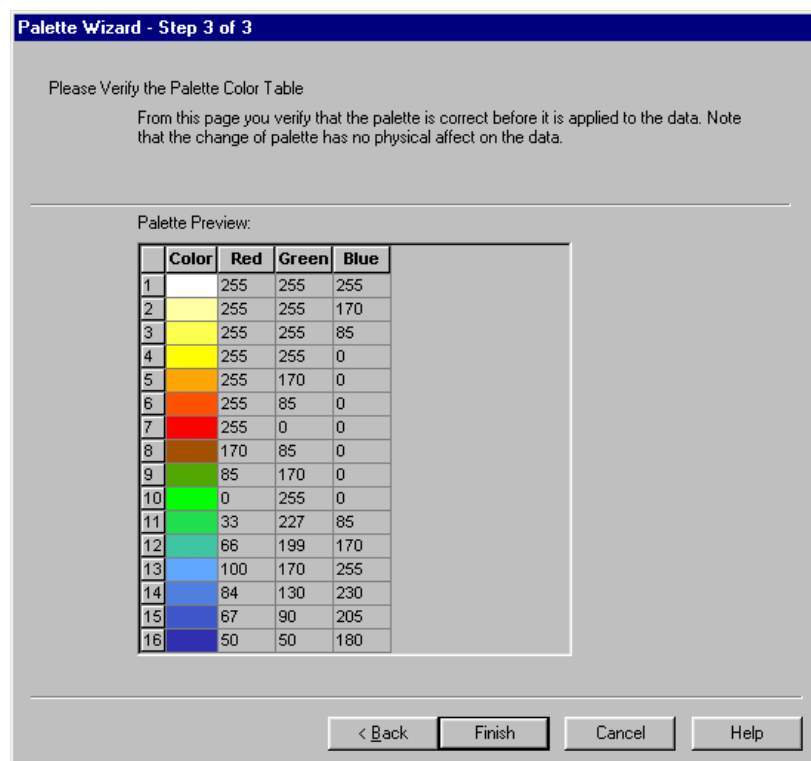
### Land Colour

If you are working with a land/water palette, then you may select the colour for land.

### Land Value

If you are working with a land/water palette, choose the value such that cells with a value larger or equal to this value will be shown as land.

## 23.3.3 Palette Wizard, step 3 of 3



In the third step you can verify that the palette that you have chosen corresponds to what you were aiming at, if so then press *Finish*, otherwise press *<Back>*.

## 23.3.4 Open

A palette that has been saved on the disk can be loaded and will immediately take effect. The palette files have the extension \*.pal.



### 23.3.5 Save

Save the present palette to disk for later re-use. The palette file should have the extension \*.pal.

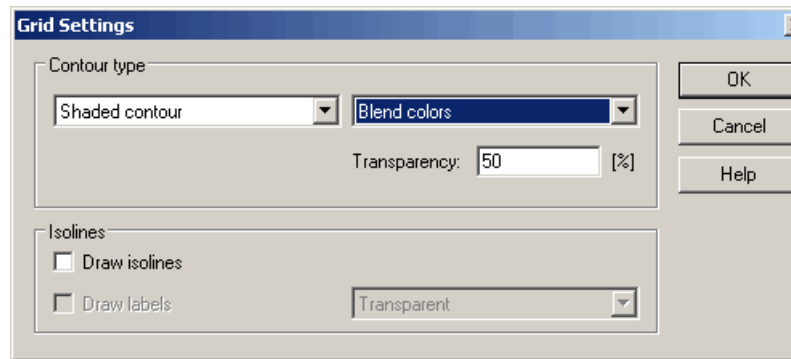
## 23.4 Overlay

A number of items can be laid over the graphical view for various purposes. Many of these options are switched on and off.

- *Grid* - overlays a grid following the grid cells in the view
- *Position of tab view* - overlays a grid mask corresponding to the tabular view to the right. Placing the pointer near the bottom end of the mask may move around this grid mask, When the pointer changes character, you can move the mask freely by keeping the left mouse button down. Drop the mask at the new location by releasing the mouse button again. The active point in the tabular view is highlighted in the mask with a distinct colour.
- *Position of perpendicular slices* - used only in connection with 3D files
- *Selection* - highlights the cells which have been selected by use of the Selection tool.
- *Color legend* - places a colour legend next to the graphical representation, the colours should be set using the Palette (p. 322).
- *Geographical net* - lines of constant longitude and latitude are placed over the graphical view
- *UTM Net...* - lines of constant Eastings and Northings are overlaid. You have to choose which UTM zone type and if it is a standard type, then you should also choose the zone number. For more information on the UTM coordinate system, see the MIKE Zero Digitizer
- *North arrow* - shows an arrow indicating the direction towards the geographical north pole



## 23.5 Grid Settings



### 23.5.1 Contour type

Here you can define the way contours are displayed in the Graphical View. The colours can be set using the Palette (*p. 322*), and a colour legend can be placed next to the grid, see Overlay (*p. 326*).

#### Box Contour

Can either be specified with or without isolines. Each grid cell is given a colour according to the value in the cell.

#### Box Contour with transparency

Used in connection with background overlays the value define the transparency.

#### Shaded Contours

The grid is covered with colours indicating the values in the cells, a smooth surface is used. This will change all land to green.

#### Shaded Contour with land

This option is similar to **Shaded Contours**, except that it covers points with land values with a colour indicative of land.

#### No Contour

Used in connection with background overlays. The bathymetry will not be shown.

### 23.5.2 Isolines

Here you can define the way isolines are displayed in the Graphical View.

**Draw Isolines**

This option can be switched on and off. A system of isolines will be placed on the grid.

**Draw labels**

Apply labels to the isolines either as transparent, opaque or framed opaque.

**23.6 Axis Annotation**

Choose here how you would like the axis annotation on the graphical view to be. The options are grid points, kilometers and meters.

**23.7 Mouse Pointer Coordinates**

When the mouse pointer is in the graphical view, the coordinates of the point and the value at the point that you are pointing at are displayed in the Status Bar.

Choose here how you would like the mouse pointer coordinates to be displayed. The options are (x,y)-coordinates, (lon,lat)-coordinates or UTM-coordinates (the UTM-zone is defined by the information in the dfs2 file).

**23.8 Fixed aspect ratio**

This option can be switched on and off. Use this option to choose between views with fixed aspect ratio or with aspect ratio adjusted to the size of the graphical view.

**23.9 Export Graphics**

The entire plot area with all plot objects within can be exported to either Clipboard, Metafile or Bitmap.

**23.10 Toolbars**

Two toolbars are particular to the Grid Editor:

- Grid Editor Tools toolbar





- Grid Editor Navigation toolbar

These toolbars provide easy and convenient access to a number of tools that are also available from the Grid Editor menu bar.

### 23.10.1 Grid Editor Tools



Pointer



Select Points (*p. 332*)



Select Lines (*p. 332*) (pull down from *Select points*)



Select Polygon (*p. 332*) (pull down from *Select points*)



Select Rectangle (*p. 332*) (pull down from *Select points*)



Select All (*p. 332*)



Select View (*p. 332*)



Select Values (*p. 333*)



Calculator (*p. 336*)



Interpolation (*p. 334*)

### 23.10.2 Grid Editor Navigation



Move one step forward in time



Rewind one step in time



Move one layer up (3D only)



Move one layer down (3D only)

Select item

## **23.11 Status Bar**

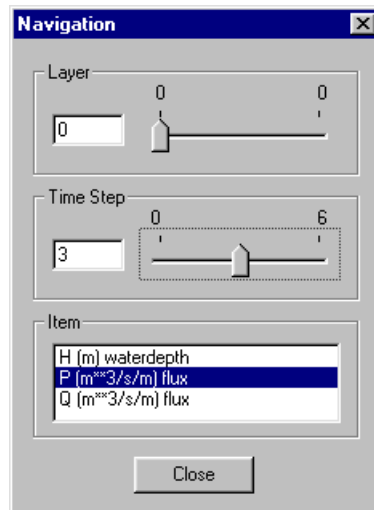
A status bar at the bottom of the application can be switched on and off.

When the pointer is in the graphical view, the status bar shows you the coordinates of the point and the value at the point that you are pointing at.



## 24 TOOLS

### 24.1 Navigation



The navigation dialog is used to position the editor at the desired layer of data (in 3D) at the desired time step and the desired item. Simply choose the wanted layer, time step and item and close. When you are working with a 3D dataset and want to switch to another plane, you should click the appropriate view at the bottom of the graphical view.

You may also conveniently use the Grid Editor Toolbars for navigating the data.

### 24.2 Go to

Using this option your active cursor will move to the minimum or maximum value in the Tabular View.

The Graphical View will not be synchronized accordingly.

### 24.3 Synchronize

**Synchronize tab to map** moves the Tabular View to the area displayed in the centre of the Graphical View.



**Synchronize map to tab** moves the Graphical View to the area displayed in the Tabular View.

## **24.4 Selection**

### **24.4.1 Select and deselect**

These menu entries under Tools are used to activate the tools for making a selection of grid cells that can then be processed in different ways. You can also use the Select a Sub-Set of Data property page that are included in many of the tools in the Grid Editor.

The Select tool is accumulative in the sense that the selections are combined into one selection. To start from scratch you should un-select the cells. You can also use the below tools in a deselect mode, such that you 'subtract' the cells from the selection. There are a number of selection tools available:

#### **Points**

Use this tool to point to individual cells that you want to select. Click once at each cell

#### **Lines**

Use this tool to select the cells along a line that you define. The cells nearest this line will be selected. The line needs not be straight, but it can consist of a number of straight segments. Click at points on the line and double click on the last point

#### **Polygon**

Use this tool to select the cells inside a polygon. Click at each corner and finish by double clicking

#### **Rectangle**

Use this tool to select the cells inside a rectangle. Place the pointer at one corner of the rectangle and pull it towards the opposite corner. When the desired rectangle is shown, then click the mouse

#### **View**

Use this tool to select the cells shown in the current view

#### **All**

Use this tool to select all cells in the active dataset



## Values

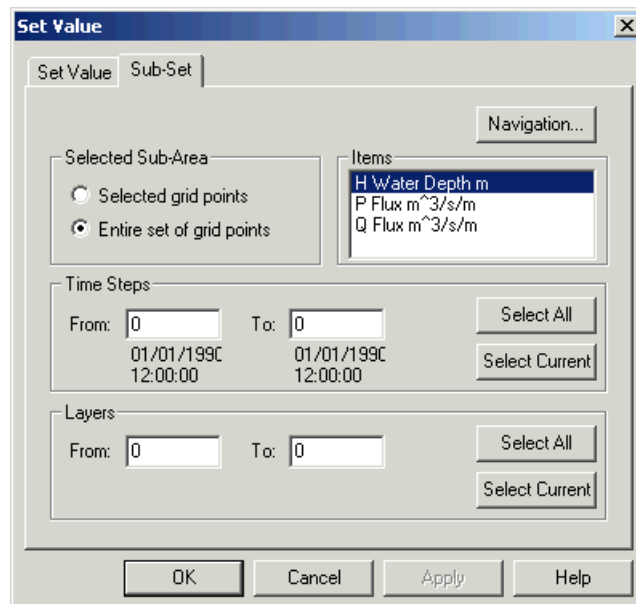
Use this tool to select the cells which do not have delete values

## Highlighted in tabular view

This select the cells highlighted in the tabular view

### 24.4.2 Select a Sub-Set of Data

Many of the tools in the Grid Editor operate on a sub-set of selected data from the dataset. The active data are defined in the 'Sub-Set' property page.



The active data should be defined in terms of:

- Sub-area: choose either the points which have been selected using the Selection option in the graphical view, or all grid points in the plane
- Items: Choose one or more of the items
- Time steps: choose a range of one or more time steps
- Layers: Choose a range of one or more layers

To change the position of the editor while the Subset page is invoked press the "Navigation..." button to get access to the Navigation dialog.



## 24.5 Interpolation

This tool can be used to interpolate in the data and fill cells with missing values.

### 24.5.1 Active Dataset

The active data are defined in the “Select a Sub-Set of Data” page.

### 24.5.2 Interpolation Settings

This parameter will determine how the interpolation will weigh the values used in the interpolation according to the distance from the interpolation point to these points. The weights can be based on the inverse of the distance, the distance squared or the distance cubed, in order to vary the relative weight given to the nearest points relative to points further away.

Land points can be ignored. This is very important when using artificial values to indicate land, because if such values are included in the interpolation this may lead to falsified results.

### 24.5.3 Search Type

#### Linear search

Depending on the type of data which are interpolated there may be an interest to restrict the search for values to base the interpolation on to either vertical or horizontal lines (i.e. along one or the other grid axis). One example hereof could be bathymetric data in the surf zone where such data could be obtained from a ship sailing along lines. Here the outcome of the interpolation could vary significantly depending on the choice of search type.

#### Area integrated

Area integrated type for interpolation performs a convolution when calculating the interpolated value. The algorithm starts its search in the current point and searched in a rectangular area around the point. The size of the rectangle is defined by the "Area side length". "Number of points" defines how many neighbouring grid points that is used in the interpolation. If e.g. 20 is specified in "Number of points" the search algorithm finds the 20 closest points inside the specified rectangle and performs an inverse distance weighted interpolation raised to the specified power chosen in interpolation settings.



## 24.6 Filter

This tool is used to perform digital filtering operations on the active data. The active data are defined in the “Select a Sub-Set of Data” page.

A number of pre-defined filters are provided. Select the filter by clicking on the stencil that you require:

|   |   |   |
|---|---|---|
| 1 | 1 | 1 |
| 1 | 1 | 1 |
| 1 | 1 | 1 |

Averaging Filter

|   |   |   |
|---|---|---|
| 1 | 1 | 1 |
| 1 | 4 | 1 |
| 1 | 1 | 1 |

Averaging Filter

|   |   |   |
|---|---|---|
| 1 | 2 | 1 |
| 2 | 4 | 2 |
| 1 | 2 | 1 |

Averaging Filter

|    |    |    |
|----|----|----|
| 0  | -1 | 0  |
| -1 | 5  | -1 |
| 0  | -1 | 0  |

Sharpening Filter

|    |    |    |
|----|----|----|
| -1 | -1 | -1 |
| -1 | 9  | -1 |
| -1 | -1 | -1 |

Sharpening Filter

The filtering process may be repeated a number of times to enhance the effect.

Land values and delete values can be included in the calculations or ignored. Typically delete values should not be included in the filtering process since they represent 'empty' cells. Instead these cells could be filled by Interpolation prior to the filtering. When filtering near land, the inclusion of land values may lead to a falsification of the data for example when an artificial constant land value has been specified in a bathymetry dataset.

## 24.7 Set Value

This tool is used for three operations on the active data:

- Set value: set all the active data to the specified value
- Add value: add the specified value to all the active data



- Multiply value: multiply all the active data by the specified value

The active data are defined in the “Select a Sub-Set of Data” page.

## 24.8 Calculator

This tool can be used to assign values to a dataset. It can be done in terms of a simple value or as an expression. If an expression that includes the current value (s) is used, no changes will be made to data points which current value equals the delete value.

### Active Dataset

The active data are defined in the “Select a Sub-Set of Data” page.

### Expression

Then construct the expression for the calculation. As you input the expression, it will be shown in the field below. There are three types that you can insert in the expression.

Operators: the four operators add, subtract, multiply and divide

Operands: j, k, l are the indices for the grid cell in the x-, y- and z-directions, respectively, and s is the cell value itself

Function: choose a mathematical function from the List of Functions below.

### 24.8.1 List of Functions

The Calculator allows you to compose your own expressions that can be used to modify values in data sets. Below you will find a table with a list of the mathematical functions you can choose between, which arguments they take and which result they return.

Table 24.1 List of functions

| Function Name | Arguments Evaluation | Evaluation                |
|---------------|----------------------|---------------------------|
| abs           | (x)                  | The absolute value of $x$ |





Table 24.1 List of functions

|              |      |   |
|--------------|------|---|
| <b>acos</b>  | (x)  | The <b>acos</b> function returns the arccosine of $x$ in the range 0 to 180 degrees. If $x$ is less than $-1$ or greater than $1$ , <b>acos</b> returns a delete value  |
| <b>asin</b>  | (x)  | The <b>asin</b> function returns the arcsine of $x$ in the range $-90$ to $90$ degrees. If $x$ is less than $-1$ or greater than $1$ , <b>asin</b> returns a delete value   |
| <b>atan</b>  | (x)  | <b>atan</b> returns the arctangent of $x$ . If $x$ is $0$ , <b>atan</b> returns $0$ . <b>atan</b> returns a value in the range $-90$ to $90$ degrees  |
| <b>atan2</b> | (xy) | <b>atan2</b> returns the arctangent of $y/x$ . If both parameters of <b>atan2</b> are $0$ , the function returns $0$ . <b>atan2</b> returns a value in the range $-180$ to $180$ degrees, using the signs of both parameters to determine the quadrant of the return value. <b>atan2</b> is well defined for every point other than the origin, even if $x$ equals $0$ and $y$ does not equal $0$ |
| <b>ceil</b>  | (x)  | The <b>ceil</b> function returns a <b>double</b> value representing the smallest integer that is greater than or equal to $x$   |
| <b>cos</b>   | (x)  | The <b>cos</b> function returns the cosine of $x$ ( $x$ in degrees)   |
| <b>cosh</b>  | (x)  | The <b>cosh</b> function returns the hyperbolic cosine of $x$   |
| <b>cube</b>  | (x)  | Returns the cube of $x$ , i.e. $x^3$  |
| <b>exp</b>   | (x)  | The <b>exp</b> function returns the exponential value of the floating-point parameter, $x$  |
| <b>floor</b> | (x)  | The <b>floor</b> function returns a floating-point value representing the largest integer that is less than or equal to $x$   |
| <b>int</b>   | (x)  | Returns the integer value of the argument $x$   |
| <b>ln</b>    | (x)  | The <b>ln</b> function returns the natural logarithm of $x$ if successful. If $x$ is negative, the function returns a delete value. If $x$ is $0$ , it returns a delete value   |



Table 24.1 List of functions

|               |      |   |
|---------------|------|---|
| <b>log10</b>  | (x)  | The <b>log10</b> function returns the base-10 logarithm of $x$ if successful. If $x$ is negative, the function returns a delete value. If $x$ is 0, it returns a delete value   |
| <b>max</b>    | (xy) | <b>max</b> returns the larger of its arguments  |
| <b>min</b>    | (xy) | <b>min</b> returns the smaller of its arguments   |
| <b>mod</b>    | (xy) | <b>mod</b> returns the floating-point remainder of $x / y$ . If the value of $y$ is 0.0, <b>mod</b> returns a delete value  |
| <b>pow</b>    | (xy) | <b>pow</b> returns the value of $x^y$   |
| <b>racos</b>  | (x)  | The <b>racos</b> function returns the arccosine of $x$ in the range 0 to $\pi$ radians. If $x$ is less than $-1$ or greater than $1$ , <b>racos</b> returns a delete value  |
| <b>rasin</b>  | (x)  | The <b>rasin</b> function returns the arcsine of $x$ in the range $-\pi / 2$ to $\pi / 2$ radians. If $x$ is less than $-1$ or greater than $1$ , <b>rasin</b> returns a delete value   |
| <b>ratan</b>  | (x)  | <b>ratan</b> returns the arctangent of $x$ . If $x$ is 0, <b>atan</b> returns 0. <b>atan</b> returns a value in the range $-\pi / 2$ to $\pi / 2$ radians   |
| <b>ratan2</b> | (xy) | <b>ratan2</b> returns the arctangent of $y/x$ . If both parameters of <b>ratan2</b> are 0, the function returns 0. <b>ratan2</b> returns a value in the range $-\pi$ to $\pi$ radians, using the signs of both parameters to determine the quadrant of the return value. <b>ratan2</b> is well defined for every point other than the origin, even if $x$ equals 0 and $y$ does not equal 0 |
| <b>rcos</b>   | (x)  | <b>rcos</b> returns the cosine of $x$ ( $x$ in radians)   |
| <b>rsin</b>   | (x)  | <b>rsin</b> returns the sine of $x$ ( $x$ in radians)   |
| <b>rtan</b>   | (x)  | <b>rtan</b> returns the tangent of $x$ ( $x$ in radians)  |
| <b>sin</b>    | (x)  | <b>sin</b> returns the sine of $x$ ( $x$ in degrees)  |
| <b>sinh</b>   | (x)  | <b>sinh</b> returns the hyperbolic sine of $x$  |
| <b>sqr</b>    | (x)  | Returns the square of $x$ , i.e. $x^2$  |

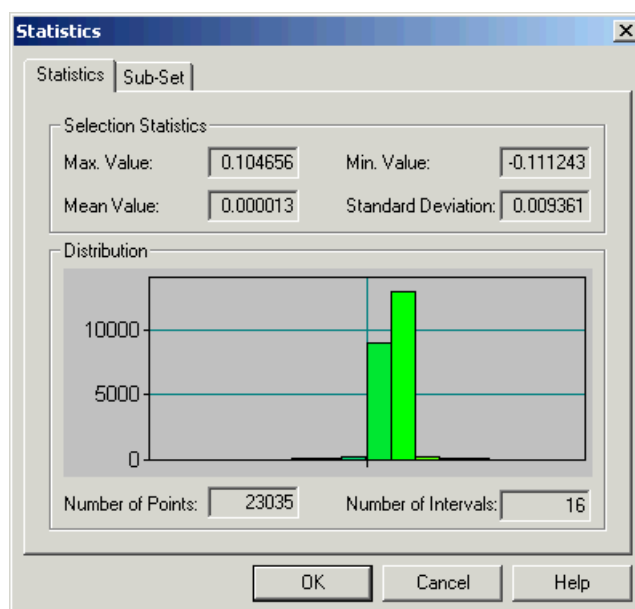


Table 24.1 List of functions

|             |            |  |
|-------------|------------|--|
| <b>sqrt</b> | <b>(x)</b> | The <b>sqrt</b> function returns the square root of $x$ . If $x$ is negative, <b>sqrt</b> returns a delete value |
| <b>tan</b>  | <b>(x)</b> | <b>tan</b> returns the tangent of $x$ ( $x$ in degrees)  |
| <b>tanh</b> | <b>(x)</b> | <b>tanh</b> returns the hyperbolic tangent of $x$  |

## 24.9 Calculate Statistics

This tool calculates the statistics of the Active Dataset and display the values in a window.



### Active Dataset

The active data are defined in the “Select a Sub-Set of Data” page.

### Statistics

The calculated values includes the minimum, maximum and mean value and the standard deviation.

Furthermore a block diagram shows the distribution of the values in the intervals defined by the Palette.



## 24.10 Copy File into Data

This tool can be used to copy data from a file into the dataset being edited.

**Copy File into Data**

File to Copy  
Filename: MIKE\_3\FlowModel\HD\Sound\Salt-North.dfs2 ...

Item Mapping

|   | Source item | Maps to                             | Target item    |
|---|-------------|-------------------------------------|----------------|
| 1 | Salinity    | <input checked="" type="checkbox"/> | Salinity [psu] |

2D to 3D Layer Mapping  
☒ Populate layer no: 4 From 0 to 40  
☐ Populate all layers

Sub-Area Position  
j-origin: 6  
k-origin: 14

Time Position  
Date origin: 1993/12/02 00:00:00  
Time step origin: 0 ☒ Interpolate

Operation  
Type: Copy

OK  
Cancel  
Help

### 24.10.1 File to Copy

Select the data file from which data should be copied.

If you edit a 2D data file you may copy data from a 2D data file.

If you edit a 3D data file you may copy data from either 2D or 3D data files.

### 24.10.2 Item Mapping

Select the items from which data should be copied and the corresponding items to which data should be copied.

You may copy several items in one process.



### **24.10.3 2D to 3D Layer Mapping**

When copying a 2D file into a 3D file you can choose to populate either one particular layer in the 3D file or all layers.

### **24.10.4 Sub-area Position**

Give the coordinates in the grid being edited where the origin of the data to be copied should be positioned. As an example, we have a grid with the dimensions (0:20,0:20) and we wish to copy data into the rectangle given by the four points (10,14), (13,14), (13,17) and (10,17). We select a file which has data in a grid (0:3,0:3) and specify j-origin=10 and k-origin=17. This will place the copied data in the desired rectangle.

### **24.10.5 Time Position**

The copied data can be placed in the dataset being edited from a given time step and forwards. You can specify either the time step or the time. If you specify the time, and that time is not coincident with a time step in the edited dataset, then the nearest time step which is shown in the time step origin edit field will be used.

### **24.10.6 Operation**

When the data is copied to the dataset, this may be done in a number of ways:

- Copy (all values are copied such that they replace the existing data in the dataset)
- Copy if target differs from delete value
- Copy if source differs from delete value
- Copy if source AND target differs from delete value
- +: the values are added to the existing data in the dataset
- -: the values are subtracted from the existing data in the dataset
- \*: the existing values are multiplied by the values in the file
- /: the existing values are divided by the values in the file

## **24.11 Crop**

This tool is used to crop (or reduce) the dataset in one or more ways.



**Crop**

**Spatial**

☒ Crop to view      min:      max:

☐ Crop to coordinates:      i: 0      117

   k: 0      138

   l: 0      0

**Item**

☒ H Water Depth m

☒ P Flux m<sup>3</sup>/s/m

☒ Q Flux m<sup>3</sup>/s/m

**Time**

From:      To:

Date: 1996/04/13 05:00:00      1996/04/13 05:04:10

Time step: 0      5

OK

Cancel

Help

### Spatial cropping

Use this option to crop the data to a smaller spatial coverage. Choose a rectangular area either as the current view or as the area described by the coordinates given by you. The data outside the selected area will be discarded.

### Item cropping

Use this option to choose the items to keep, the rest of the items are discarded.

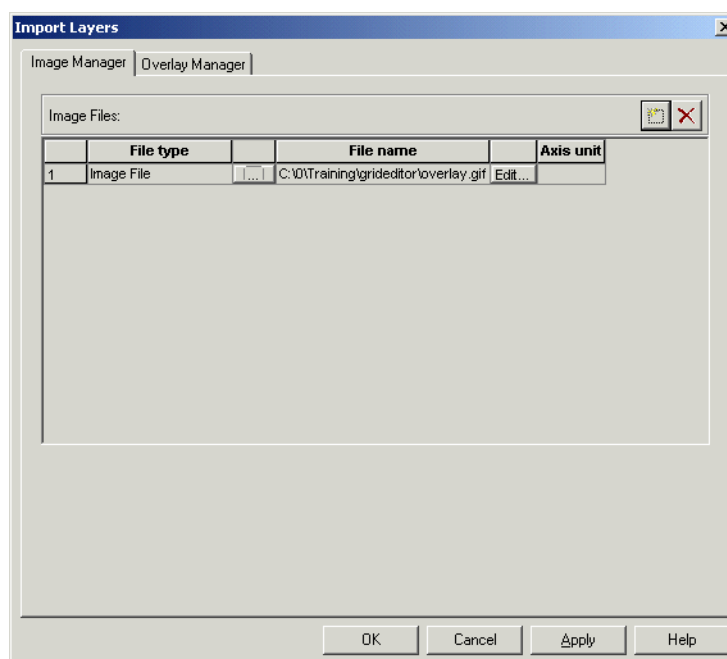
### Time cropping

Use this option to discard time steps in the dataset. The data outside the chosen interval will be discarded. The time interval can be selected in terms of the time or the time step number.



## 25 DATA OVERLAY

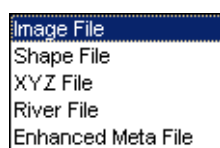
### 25.1 Image Manager



In the Image Manager you specify the overlay file name and type.

If you want to save the overlay image together with the specific grid file, you can use the Grid State Format (*p. 308*)

#### Image Types



Select between:

- Image Files: BMP, JPG, GIF
- Shape File: From ArcInfo: SHR
- XYZ File: Digitized ascii file with x,y and z co-ordinates: XYZ



- River File: MIKE11 File: nwk11
- Enhanced Meta File: EMF

Image files can be used as background if they are grid aligned and cover the same area as defined by the grid.

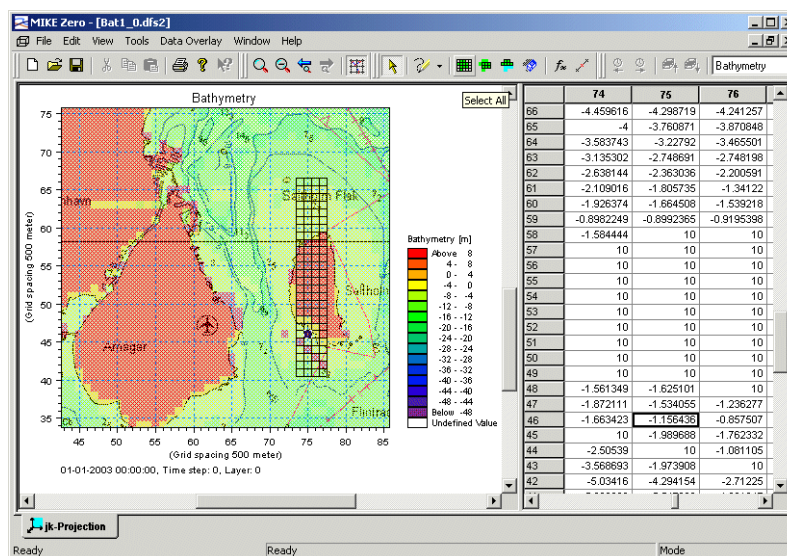
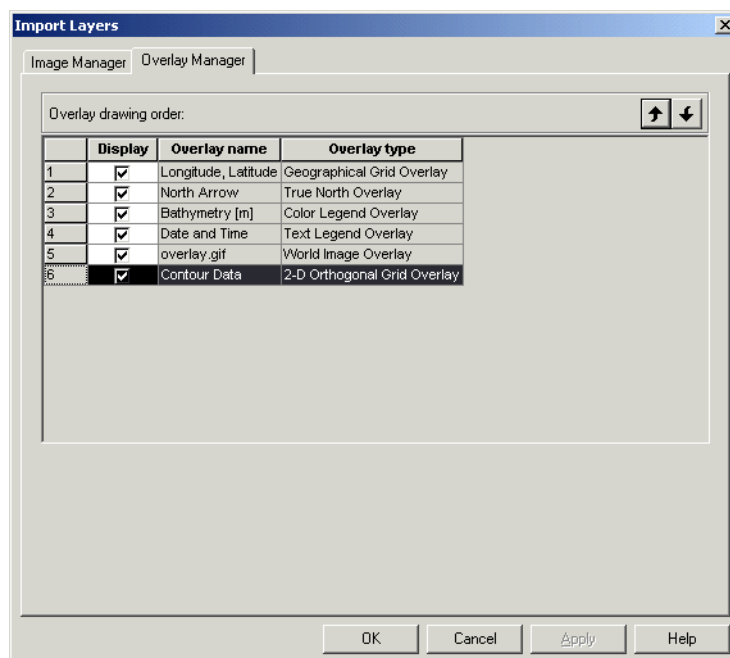


Figure 25.1 Example of background layer





## 25.2 Overlay Manager



Specify which layer to display and the display order. Remember to adjust the transparency level in Grid Settings (*p.* 327).





*Phi Software*

## **MIKE ZERO OPTIONS**



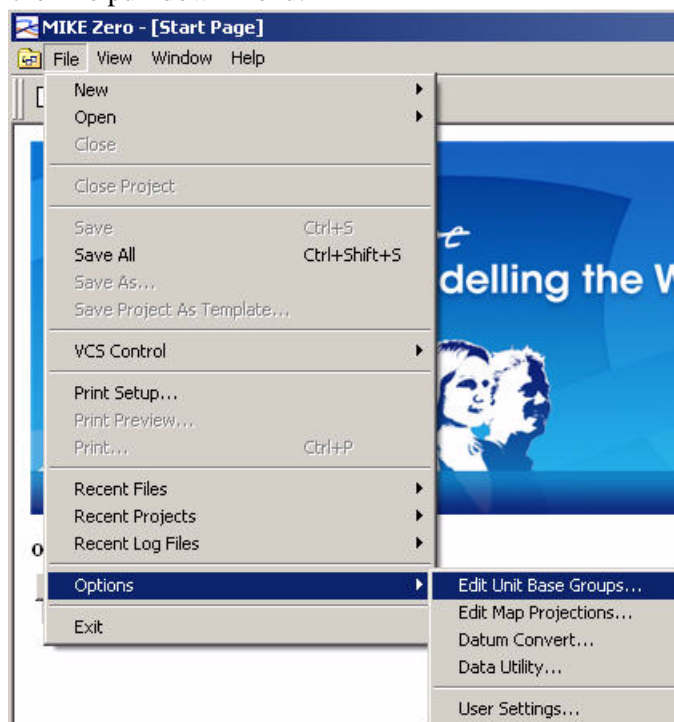


## 26 EUM DATA UNITS

All MIKE Zero products use a standard library of data units, called the Engineering Unit Management (EUM) library. This allows you to change the displayed units for any value that is included in the library.

Every parameter in MIKE SHE has been added to the EUM library and to change the displayed unit, you must know the EUM Data Type. In most cases, the EUM Data Type is displayed in the fly-over text when you put your mouse cursor in the text field. Alternatively, all items in the on-line help (F1) list the EUM Data Type in the table at the beginning of the section.

To change the display units of any EUM Data Type, you must close all open documents and then select 'Options/Edit Unit Base Groups...' from the File pull down menu.



When you select this menu item, the Unit Base Group Editing dialogue appears. By default all of the data units for each active module are displayed. For a clearer overview of the data types, close all of the model engines that are not relevant.

Next select the data item that you want to change the units of. Then select the new units from the combobox list of available units.



After you have changed the data units, click 'Save and Close'. This saves your changes to the default Unit Base Groups (.ubg) file:

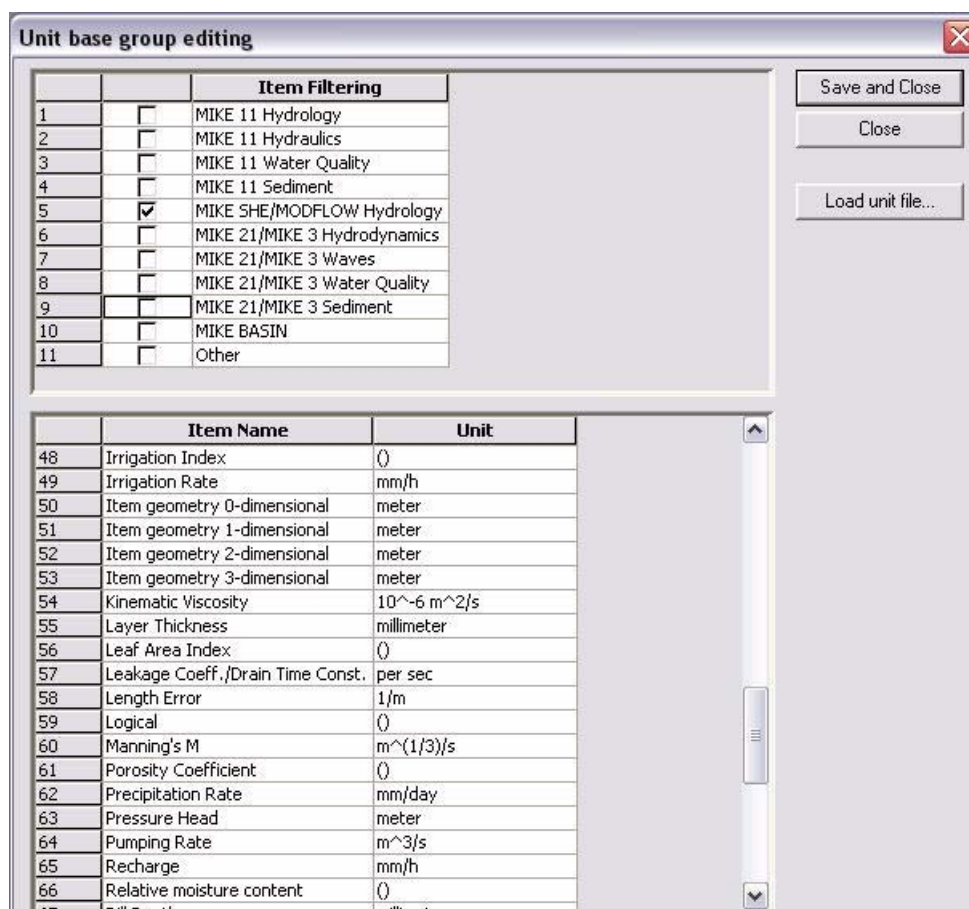
C:\Program Files\Common Files\DHI\MIKEZero\MIKEZero.ubg

which is read every time you open a model.

**Note!** If you have already added data to your model, changing the Unit Base Group will not convert any of your data. This process simply changes the displayed units in the user interface and the conversion factors used to make the input files internally consistent.

In some cases the relevant data item name is not clear, as there may be several data items with similar names. This is more likely to occur if several modules are selected at the same time. To find out which data item is correct, close the dialogue and re-open your model. Then either move the mouse to the relevant textbox, where a fly-over text box should appear telling you what is the relevant data type for this field. Alternatively, for gridded data, you can use the Create button to create a data file and then notice the data type that is displayed in the dialogue.

Finally, occasionally, you may find that the data unit that you are looking for is not available. In this case, contact your local Technical Support Cen-



tre, who should forward your request to the developer for inclusion in the next release.

## 26.1 Changing from SI to Imperial (American) data units.

The default Unit Base Groups (.ubg) file,

C:\Program Files\Common Files\DHI\MIKEZero\MIKEZero.ubg

is read every time you open a model.

In the same directory there are two standard Unit Base Group files:

MIKEZero\_Default\_Units.ubg

MIKEZero\_US\_Units.ubg



The first is the default file and contains standard SI units for all data items in all of the MIKE Zero products. The second contains standard Imperial (US) units for most data items in all of the MIKE Zero products.

To change the display units for all of your data items to Imperial units, load the MIKEZero\_US\_Units.ubg file, Save and Close the dialogue and then reopen your model.

If you want to change individual data items to SI or Imperial, you can change the items individually. Then use the Save and Close button to save your changes back to the MIKEZero.ubg file. If you want to create special unit versions, then you can copy the MIKEZero.ubg to a different file name and reload it.

## **26.2 Restoring the default units**

You can return to your default unit specification at any time, by Loading either of the default .ubg files:

MIKEZero\_Default\_Units.ubg

MIKEZero\_US\_Units.ubg

which are found in the

C:\Program Files\Common Files\DHI\MIKEZero\

directory.

**Note!** If you want to save any of your model specific changes, then you should first save the MIKEZero.ubg to a new name.

## **26.3 Changing the EUM data type of a Parameter**

When you create a .dfs0 or .dfs2 parameter file, you must also define the EUM data type for each parameter in the file. When you assign a .dfs0 or a .dfs2 file to a parameter value, then MIKE SHE automatically verifies that the correct EUM data type is being used. If the wrong data type is present then you will not be able to select OK in the file browser dialogue.

For example, in the following set of dialogues, an Evapotranspiration time series was selected instead of the correct Precipitation time series file





The first error is in the Select Item tab, where there is a message that no Valid Items are found.

| Select Precipitation Rate | Item           |
|---------------------------|----------------|
|                           | No valid items |

To find out why there is no valid items, you should look in the Constraints Info tab

| Status | Constraints                                 |
|--------|---|
|        | Item Request No. 1                          |
| ✗      | Number of dimensions = 0                    |
|        | Item type=Precipitation Rate                |
|        | Validation of Data Period                   |
| ✓      | Start Date of Data Period before 02/01/1990 |
| ✓      | End Date of Data Period after 01/11/1995    |

Here you can see that the Item type is supposed to be Precipitation Rate, but this constraint has failed.

To find out what the Item Type of the selected file is, look at the Item Info tab,

|   | Item Name                    | Item Type               | Unit   |
|---|------------------------------|-------------------------|--------|
| 1 | Potential EvapoTranspiration | Evapotranspiration Rate | mm/day |

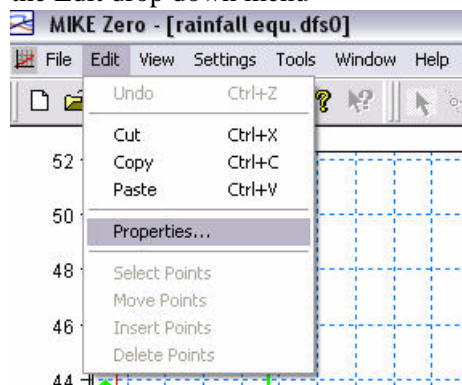
where you can see that the current Item Type is Evapotranspiration Rate.



The next two sections outline how to change the EUM Type of an existing file.

### 26.3.1 Changing the EUM Type of a .dfs0 Parameter

To change the EUM Data Type of a parameter in a .dfs0 file, open the time series in the Time Series Editor and then select the Properties... item from the Edit drop down menu



This opens the item properties dialogue

**File Properties**

General Information

Title:

Axis Information

Axis Type:

Start Time:

Time Step:  [days]  
 [hour:min:sec]  
 [fraction of sec.]

No. of Timesteps:  Axis Units:

Item Information

|   | Name  | Type               | Unit   |               |
|---|-------|--------------------|--------|---------------|
| 1 | 10258 | Precipitation Rate | mm/day | Mean Step Acc |
| 2 | 10380 | Precipitation Rate | mm/day | Mean Step Acc |
| 3 | 10401 | Precipitation Rate | mm/day | Mean Step Acc |
| 4 | 10402 | Precipitation Rate | mm/day | Mean Step Acc |

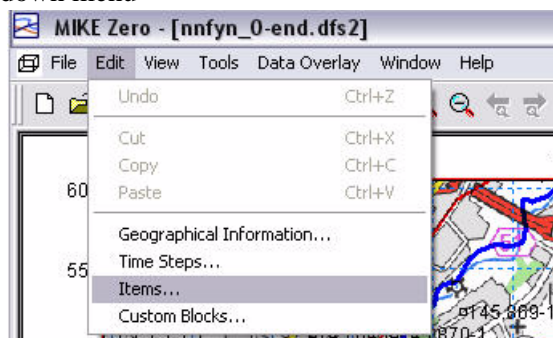
Insert Append Delete Item Filtering...



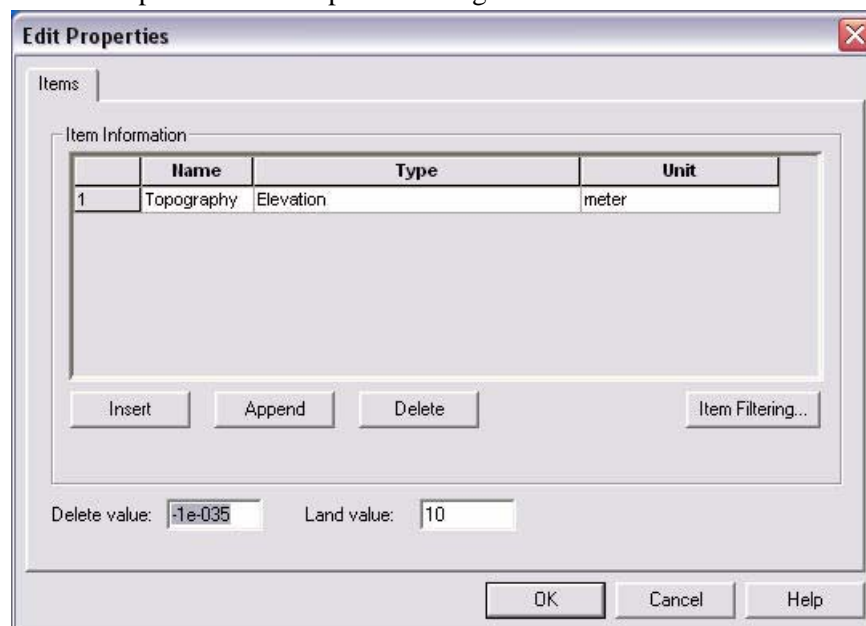
where you can change the EUM Type and the EUM Unit that is assigned for each time series in the file.

### 26.3.2 Changing the EUM Type of a .dfs2 Parameter

To change the EUM Data Type of a parameter in a .dfs2 file, open the grid file in the Grid Editor and then select the Items... item from the Edit drop down menu



This will open the Edit Properties dialogue for the Grid Editor



where you can change the EUM Type and the associated data EUM Unit of the item.





*Phi Software*

**AUTOCAL**





## 27 AUTO CALIBRATION TOOL

AUTOCAL is a generic tool for performing automatic calibration, parameter optimisation, sensitivity analysis and scenario management of the numerical modelling engines under MIKE Zero. It interfaces the modelling engines using the MIKE Zero PFS (parameter files system) format for model input and the DFS (data file system) format for model output. AUTOCAL can also be linked to other modelling engines by interfacing ASCII model input files and creating a bridge between the model output and the DFS file format.

The core of the AUTOCAL tool consists of the following steps:

- 1 Provision of a set of model parameter values to the numerical model to be used in a specific model run.
- 2 Execution of the simulation model using the specified set of model parameter values.
- 3 Calculation of statistical performance measures of the model output.

The model parameter interface is made via a so-called template file. This file is simply a replica of the model input file in which parameter identification tags are placed at the locations where the numerical values of the model parameters are given. Thus, when AUTOCAL has to provide a new set of model parameters to the model, the parameter identification tags in the template file are substituted by the numerical values and saved in the model input file to be used when running the model. It is possible to manipulate model parameters in different model input files. For each model input file a corresponding template file is defined.

AUTOCAL allows parameters to be defined as functions of other parameters that are manipulated in the AUTOCAL run. In this case general equations can be specified that defines the relations between the dependent model parameter and the other parameters. AUTOCAL provides the set of independent parameter values and the dependent parameters are then calculated automatically from these values using the defined equations. Both dependent and independent parameter values are then substituting their corresponding parameter identification tags in the template files. Besides the standard arithmetic operators, the equation tool includes a large number of mathematical functions.

A parameter may also be defined as a constant. In this case the constant value defined in AUTOCAL is substituting the corresponding parameter identification tag in the template file. This feature is especially useful when a sensitivity analysis is performed prior to the parameter optimisa-



tion. In this case the sensitivity analysis typically includes a long list of parameters. From the results of the sensitivity analysis the most sensitive parameters are retained in the subsequent parameter optimisation. Instead of making a new AUTOCAL setup, the same setup as used in the sensitivity analysis can be applied simply by setting the insensitive parameters to constant values.

A model simulation in AUTOCAL can be defined as a sequence of individual model runs. AUTOCAL supports execution of all the MIKE Zero model engines. In this case the model simulation setup file is given as input. In addition, AUTOCAL enables execution of any executable (.exe) or batch (.bat) file. In this case the executable as well as the corresponding list of arguments are given as input.

Whether AUTOCAL is used for parameter optimisation, sensitivity analysis or scenario management, the performance of the model simulation given the specified parameter set should be assessed. This is done by calculating statistical performance measures. These measures are typically comparison statistics that compare measurements or, in general, target values with corresponding simulated values.

For calculating the comparison statistics AUTOCAL requires that simulation results and corresponding observations are given as time series in DFS0 files. If the output from a model engine is not explicitly given in DFS0 format, a processing of simulation results is required to transform the simulation results at measurement locations into DFS0 format. This post-processing is then part of the sequence of model runs defined in AUTOCAL. Typically post-processing is required for MIKE 11 result files (.res11) and for models that provide results in 1D, 2D or 3D grid files (.dfs1, .dfs2 and .dfs3). For application of AUTOCAL with modelling engines that are not included in MIKE Zero the COM components included in the MIKE Objects Time Series Package can be used to create DFS0 time series files from the model output files.

Before setting up AUTOCAL, the simulation model should be properly tested. At least one model run should be performed to create the output files that are needed in the AUTOCAL setup.

## **27.1 New AUTOCAL Dialog**

To create a new AUTOCAL dialog from MIKE Zero choose:

File | New | MIKE Zero | Auto Calibration





See also Figure 27.1.

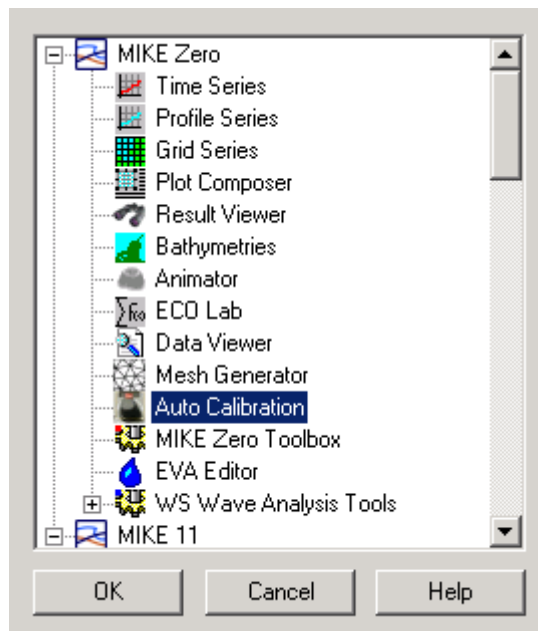
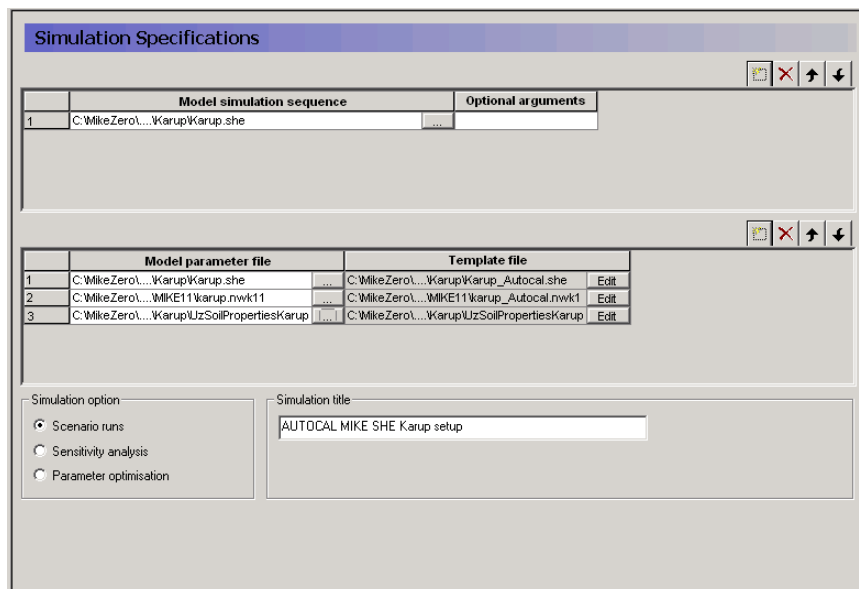


Figure 27.1 New Auto Calibration dialog

## 27.2 Simulation Specifications

On the *Simulation Specifications* page (Figure 27.2), the model simulation sequence, the model parameter files, and the simulation option are specified.



The dialog box is titled "Simulation Specifications". It contains three main sections:

- Model simulation sequence:** A table with two columns: "Model simulation sequence" and "Optional arguments". It contains one row with the value "C:\MikeZero\...\Karup\Karup.she".
- Model parameter file and Template file:** Two side-by-side tables. The "Model parameter file" table has three rows with values: "C:\MikeZero\...\Karup\Karup.she", "C:\MikeZero\...\MIKE11\karup.nwk11", and "C:\MikeZero\...\Karup\UzSoilPropertiesKarup". The "Template file" table has three rows with values: "C:\MikeZero\...\Karup\Karup\_Autocal.she", "C:\MikeZero\...\MIKE11\karup\_Autocal.nwk1", and "C:\MikeZero\...\Karup\UzSoilPropertiesKarup". Each row in the template file table has an "Edit" button.
- Simulation option:** A group box containing three radio buttons: "Scenario runs" (selected), "Sensitivity analysis", and "Parameter optimisation".
- Simulation title:** A text box containing the value "AUTOCAL MIKE SHE Karup setup".

Figure 27.2 Simulation Specifications page

### 27.2.1 Model simulation sequence

A model simulation in AUTOCAL can be defined as a sequence of individual model runs. As an example, a modelling sequence may consist of a hydrodynamic (HD) model simulation followed by an advection dispersion (AD) model simulation that uses the HD output. In this case two individual model runs must be defined in the *Model simulation sequence* table, including the HD run in the first row and the AD run in the second row. Another typical example is that a model simulation is followed by a post-processing model run to extract the simulation results at measurement points and save these in DFS0 files.

AUTOCAL supports execution of the MIKE Zero numerical engines shown in Table 27.1. In this case the model simulation file for the numerical engine (e.g. a .sim11 file for a MIKE 11 run and a .m21 file for a MIKE 21 run) has to be specified in the *Model simulation sequence* column. AUTOCAL also supports execution of any executable (.exe) and batch (.bat) file. In this case the name of the executable file or batch file is given in the *Model simulation sequence* column. If the executable file



requires some arguments, such as an input file name and some simulation options, these are written in the *Optional arguments* column.

Table 27.1 MIKE Zero numerical engines supported by AUTOCAL

| Simulation model                   | File extension |
|------------------------------------|----------------|
| MIKE SHE Flow Model                | .she           |
| MIKE SHE Transport Setup File      | .tsf           |
| MIKE 11 Simulation                 | .sim11         |
| MIKE 21 Flow Model                 | .m21           |
| MIKE 21 Flow Model FM              | .m21fm         |
| MIKE 21 Spectral Waves FM          | .sw            |
| MIKE 21 Boussinesq Waves           | .bw            |
| MIKE 21 Nearshore Spectral Waves   | .nsw           |
| MIKE 21 Elliptic Mild-Slope Waves  | .ems           |
| MIKE 21 Parabolic Mild-Slope Waves | .pms           |
| MIKE 21 Non-Cohesive Sediments     | .st2           |
| MIKE 21C Flow Model                | .m21c          |
| MIKE 3 Flow Model                  | .m3            |
| MIKE 3 Flow Model FM               | .m3fm          |
| MIKE 3 Particle/Spill Analysis     | .npa           |
| MIKE Zero Toolbox                  | .mzt           |

For post-processing of model results AUTOCAL supports extraction of time series from 1D, 2D and 3D result files using the MIKE Zero Toolbox. By using this toolbox the extraction of time series from DFS1, DFS2, or DFS3 model result files corresponding to the measurement points can be specified. The toolbox is saved as a .mzt file and can subsequently be executed by AUTOCAL by specifying the .mzt file in the *Model simulation sequence* table.

In MIKE SHE and MIKE 11 HD it is possible to specify time series output that can be used directly by AUTOCAL. For extraction of time series of other MIKE 11 variables from a MIKE 11 result file the `res11read` applica-



tion program can be used. The res11read executable is located in the MIKEZero\bin directory. The *Model simulation sequence* column should then include:

```
C:\Program Files\DHI\MIKEZero\bin\res11read.exe
```

with the *Optional arguments*:

```
-somerSpecFileName -MakeDfs0 -silent Res11FileName  
Dfs0FileName
```

where SpecFileName is the file name of the file that contains specifications of the data to be extracted from the .res11 result file (item, chainage and rivername), Res11FileName is the file name of the .res11 result file, and Dfs0FileName is the file name of the DFS0 file where the extracted time series are saved (note that this file name should not include the .dfs0 extension). Note that the full path name should be used for the specified files. Instead of using the executable option in the *Model simulation sequence* a batch file can be defined that contains the res11read command line.

More information about the res11read application program is given in the MIKE 11 User Guide, Appendix B.

### 27.2.2 Model parameter files

In the *Model parameter files* table the files containing the model parameters to be manipulated by AUTOCAL should be specified. AUTOCAL supports manipulation of parameters in the model parameter files associated with the numerical engines given in Table 27.1. The model parameter file types are shown in Table 27.2. For some simulation models only one model parameter file is associated, whereas other models (e.g. MIKE 11 and MIKE SHE) include several model parameter files. AUTOCAL also supports manipulation of parameters in ASCII files that do not follow the MIKE Zero PFS format. In the *Model parameter files* table all files that include parameters to be manipulated in the AUTOCAL run should be specified.

Table 27.2 Model parameter files supported by AUTOCAL

| Model parameter file              | File extension |
|-----------------------------------|----------------|
| MIKE SHE Flow Model               | .she           |
| MIKE SHE UZ Soil Properties       | .uzs           |
| MIKE SHE ET Vegetation Parameters | .etv           |



Table 27.2 Model parameter files supported by AUTOCAL

| Model parameter file                     | File extension |
|--|----------------|
| MIKE 11 RR Parameters                    | .rr11          |
| MIKE 11 HD Parameters                    | .hd11          |
| MIKE 11 River Network Parameters         | .nwk11         |
| MIKE 11 AD Parameters                    | .ad11          |
| MIKE 11 WQ Parameters                    | .wq11          |
| MIKE 11 ST Parameters                    | .st11          |
| MIKE 11 ECOLAB                           | .ecolab11      |
| MIKE 11 Cross Section Resistance Numbers | .xns11r        |
| MIKE 21 Flow Model                       | .m21           |
| MIKE 21 Flow Model FM                    | .m21fm         |
| MIKE 21 Spectral Waves FM                | .sw            |
| MIKE 21 Boussinesq Waves                 | .bw            |
| MIKE 21 Nearshore Spectral Waves         | .nsw           |
| MIKE 21 Elliptic Mild-Slope Waves        | .ems           |
| MIKE 21 Parabolic Mild-Slope Waves       | .pms           |
| MIKE 21 Non-Cohesive Sediments           | .st2           |
| MIKE 21C Flow Model                      | .m21c          |
| MIKE 3 Flow Model                        | .m3            |
| MIKE 3 Flow Model FM                     | .m3fm          |
| MIKE 3 Particle/Spill Analysis           | .npa           |
| ECOLAB                                   | .ecolab        |
| General ASCII input files                | *.*            |

When a model parameter file is selected, a corresponding template file is automatically created (a “\_Autocal” is added to the file name). This template file is simply a replica of the model parameter file which is used for placing parameter identification tags at the locations where the values of the model parameters to be manipulated in AUTOCAL are given. From



AUTOCAL the template file can be directly edited. In the case a MIKE Zero PFS file is selected the corresponding editor is opened, whereas a text editor (Notepad) is opened in the case of a general ASCII input file. As identification tags, reserved floating point values are used, consisting of the values 1.01e-35, 1.02e-35, ..., 9.99e-35. Thus, in the model editor the identification tag is placed in the field where the numerical value of the model parameter is given. Individual parameters must be given unique identification tags.

If several model parameters should be given the same parameter value in AUTOCAL, the same identification tag can be given to these model parameters. Alternatively, individual identification tags can be given and the parameters may be set to be identical using the dependent parameter option on the *Model Parameters* page.

When the template file is created and parameter identification tags are entered, a table with the model parameters is automatically created on the the *Model Parameters* page. Afterwards, when the template file is edited and identification tags are added or deleted, the *Model Parameters* page needs to be reset.

To reset the *Model Parameters* page choose:

Run | Reset Model Parameters Table

See also Figure 27.3 below.

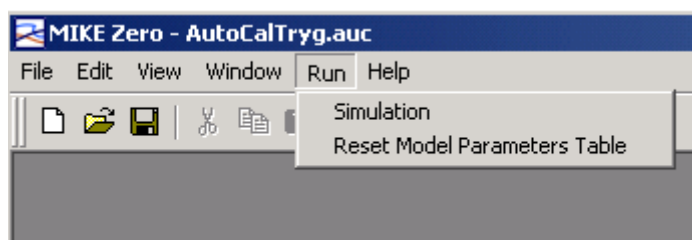


Figure 27.3 Reset Model Parameters Table

When running AUTOCAL, a second template file is created (.tpl is added to the file name) that is used internally by AUTOCAL.

It should be noted that the use of identification tags for parameter values in the MIKE Zero editors requires that the units of the chosen parameters are the default SI units. Thus, if other units have been selected these units should be changed to the default units before setting up AUTOCAL.



### 27.2.3 Simulation option

AUTOCAL supports three different simulation options:

- Scenario runs
- Sensitivity analysis
- Parameter optimisation

Depending on the choice a corresponding property page is shown in the tree view.

### 27.2.4 Simulation title

Optionally a simulation title can be specified.

## 27.3 Model Parameters

On the *Model Parameters* page (Figure 27.4) the properties of the model parameters that are defined in the model parameter files are specified. The parameter table is automatically created by AUTOAL based on the specifications given in the template files on the *Simulation Specifications* page.

| Model Parameters |           |             |                |               |             |             |                |              |           |         |
|------------------|-----------|-------------|----------------|---------------|-------------|-------------|----------------|--------------|-----------|---------|
|                  | ID value  | Name        | Parameter type | Initial value | Lower bound | Upper bound | Transformation | Equation     | Keyword   | Comment |
| 1                | 1.01e-035 | Kh_Unit2    | Variable       | 0.0001        | 1e-005      | 0.001       | Logarithmic    |              | Karup_Au  |         |
| 2                | 1.02e-035 | Kv_Unit2    | Dependent      | 0             | 0           | 0           | Logarithmic    | 0.1*Kh_Unit2 | Karup_Au  |         |
| 3                | 1.03e-035 | Kh_Unit4    | Variable       | 1e-005        | 1e-006      | 0.0001      | Logarithmic    |              | Karup_Au  |         |
| 4                | 1.04e-035 | Kv_Unit4    | Dependent      | 0             | 0           | 0           | Logarithmic    | 0.1*Kh_Unit4 | Karup_Au  |         |
| 5                | 1.05e-035 | DrainLevel  | Variable       | -1            | -1.3        | -0.8        | Real           |              | Karup_Au  |         |
| 6                | 1.06e-035 | DrainConst  | Variable       | 1e-007        | 1e-008      | 1e-006      | Logarithmic    |              | Karup_Au  |         |
| 7                | 1.07e-035 | LeakCoef    | Variable       | 1e-007        | 1e-008      | 1e-006      | Logarithmic    |              | karup_Aut |         |
| 8                | 1.09e-035 | Alpha_Soil1 | Variable       | 0.05          | 0.01        | 0.1         | Logarithmic    |              | UzSoilPro |         |
| 9                | 1.08e-035 | Ks_Soil1    | Variable       | 1e-005        | 1e-006      | 0.0001      | Logarithmic    |              | UzSoilPro |         |
| 10               | 1.11e-035 | Alpha_Soil2 | Variable       | 0.05          | 0.01        | 0.1         | Logarithmic    |              | UzSoilPro |         |
| 11               | 1.1e-035  | Ks_Soil2    | Constant       | 1e-005        | 0           | 0           | Logarithmic    |              | UzSoilPro |         |

Figure 27.4 Model Parameters page

The following properties are specified in the table:

**ID value**

The ID value is the identification tag value given for the parameter in the template file.

**Keyword/Line no**

The keyword is an identification string that shows the location of the parameter in the PFS model input file. The first part of the string is the name of the template file. This is followed by the hierarchy of PFS sections separated by dots. The last part of the string is the PFS keyword. If the parameter file is not a MIKE Zero PFS file, the line No. where the parameter is located is shown.

For each parameter, the following has to be specified:

**Name**

The user must specify a unique name for each parameter. This name must not include white spaces. In addition, if the parameter is included as an independent parameter in an equation, arithmetic symbols and function names used by the equation parser must not be used as part of the parameter name.

**Parameter type**

The parameter may be defined as either a *Variable* parameter, a *Constant* parameter or a *Dependent* parameter. A variable parameter is a parameter that is changed by AUTOCAL according to the chosen simulation option. For a variable parameter the *Initial value*, *Lower bound* and *Upper bound* need to be defined. A constant parameter is set to the value defined in the *Initial value* field. A dependent parameter is defined as a function of the other parameters. In this case the *Equation* must be specified.

**Initial value**

The *Initial value* is the value used by AUTOCAL for performing a single scenario run. If the *Local sensitivity analysis* option is chosen, the sensitivity coefficients are evaluated around the initial parameter set.

**Lower bound**

The *Lower bound* specifies the lower limit of the feasible parameter values in the parameter optimisation.

**Upper bound**

The *Upper bound* specifies the upper limit of the feasible parameter values in the parameter optimisation.





### Transformation

The parameter may be used in AUTOCAL as its native value by setting the transformation field to *Real* or as its logarithmic transformed value by setting the transformation field to *Logarithmic*. A logarithmic transformation is generally recommended if the feasible range of the parameter varies over orders of magnitude.

### Equation

If a parameter is defined as a dependent parameter, an equation must be given to define the parameter as a function of the available variable parameters. AUTOCAL uses an equation parser that supports the general arithmetic operators (+, -, \*, /) as well as a number of mathematical functions. The list of available mathematical functions is given in Table 27.3.

Table 27.3 Mathematical functions used by the equation parser (*X* and *Y* are variable names)

| Syntax   | Function  |
|----------|---|
| SQR(X)   | Square function   |
| SQRT(X)  | Square root function  |
| SIN(X)   | Sinus function. SIN returns the sine of the angle X in radians            |
| COS(X)   | Cosinus function. COS returns the cosine of the angle X in radians        |
| TAN(X)   | Tangent function. TAN returns the tangent of the angle X in radians       |
| COTAN(X) | Cotangent function. COTAN returns the cotangent of the angle X in radians |
| ATAN(X)  | ArcTangent function   |
| EXP(X)   | Exponential function  |
| LN(X)    | Natural logarithmic function  |
| LOG(X)   | 10 based logarithmic function   |
| SINH(X)  | Sinus Hyperbolic function   |
| COSH(X)  | Cosinus Hyperbolic function   |



Table 27.3 Mathematical functions used by the equation parser (X and Y are variable names)

| Syntax      | Function   |
|-------------|--|
| INTPOW(X,Y) | The INTPOW function raises X to an integer power Y, e.g. $\text{INTPOW}(2, 3) = 8$ . Note that the result of $\text{INTPOW}(2, 3.4) = 8$ as well |
| POW(X,Y)    | The POW function raises X to any power Y   |
| ABS(X)      | Absolute value   |
| SIGN(X)     | SIGN(X) returns -1 if $X < 0$ ; +1 if $X > 0$ , 0 if $X = 0$   |
| TRUNC(X)    | Discards the fractional part of a number, e.g. $\text{TRUNC}(3.2)$ is 3  |
| MIN(X,Y)    | Minimum of X and Y, e.g. $\text{MIN}(2,3)$ is 2  |
| MAX(X,Y)    | Maximum of X and Y, e.g. $\text{Max}(2,3)$ is 3  |

As an example, suppose Y is to be expressed as a function of the variables X1, X2 and X3 as 5 times variable X1 minus the square of variable X2 plus 2 times the natural logarithm of X3, the *Equation* field for Y should be written:

$5 * X1 - \text{SQR}(X2) + 2 * \text{LN}(X3)$

#### Comment

Optionally a comment can be written for the parameter.

## 27.4 Objective Functions

On the *Objective Functions* page (Figure 27.5) the properties for calculation of comparison statistics are specified. The basic statistics used by AUTOCAL are the *Output measures* that include a single comparison statistic between an observed and a simulated time series. These basic measures can then be aggregated into different *Objective functions*, for instance according to spatial location, type of variable, or type of statistic. Finally, the defined objective functions are aggregated into a single statistic that is used by the optimisation algorithm.



**Objective Functions**

Evaluation period  
 Start date: 01-01-1971 00:00:00  
 End date: 01-01-1974 00:00:00

Aggregation of objective functions  
 Transformation to a common distance scale

Objective functions

|   | Name        | Function type           | Weight |
|---|-------------|-------------------------|--------|
| 1 | RMSE_Wells  | Weighted sum of squares | 1      |
| 2 | AE_Wells    | Weighted sum            | 1      |
| 3 | RMSE_Runoff | Weighted sum of squares | 1      |

Output measures

|    | Name        | Output file              | Item name     | Observation file       | Item name   | Statistic type | Weight | Function name |
|----|-------------|--------------------------|---------------|------------------------|-------------|----------------|--------|---------------|
| 1  | RMSE_Well5  | C:\MikeZero\...\Karup\Ka | Well 5        | C:\MikeZero\...\TIME\H | Obs Well 5  | RMSE           | 1      | RMSE_Wells    |
| 2  | RMSE_Well9  | C:\MikeZero\...\Karup\Ka | Well 9        | C:\MikeZero\...\TIME\H | Obs Well 9  | RMSE           | 1      | RMSE_Wells    |
| 3  | RMSE_Well12 | C:\MikeZero\...\Karup\Ka | Well 12       | C:\MikeZero\...\TIME\H | Obs Well 12 | RMSE           | 1      | RMSE_Wells    |
| 4  | RMSE_Well21 | C:\MikeZero\...\Karup\Ka | Well 21       | C:\MikeZero\...\TIME\H | Obs Well 21 | RMSE           | 1      | RMSE_Wells    |
| 5  | RMSE_Well22 | C:\MikeZero\...\Karup\Ka | Well 22       | C:\MikeZero\...\TIME\H | Obs Well 22 | RMSE           | 1      | RMSE_Wells    |
| 6  | AE_Well5    | C:\MikeZero\...\Karup\Ka | Well 5        | C:\MikeZero\...\TIME\H | Obs Well 5  | Avg. Error     | 1      | AE_Wells      |
| 7  | AE_Well9    | C:\MikeZero\...\Karup\Ka | Well 9        | C:\MikeZero\...\TIME\H | Obs Well 9  | Avg. Error     | 1      | AE_Wells      |
| 8  | AE_Well12   | C:\MikeZero\...\Karup\Ka | Well 12       | C:\MikeZero\...\TIME\H | Obs Well 12 | Avg. Error     | 1      | AE_Wells      |
| 9  | AE_Well21   | C:\MikeZero\...\Karup\Ka | Well 21       | C:\MikeZero\...\TIME\H | Obs Well 21 | Avg. Error     | 1      | AE_Wells      |
| 10 | AE_Well22   | C:\MikeZero\...\Karup\Ka | Well 22       | C:\MikeZero\...\TIME\H | Obs Well 22 | Avg. Error     | 1      | AE_Wells      |
| 11 | RMSE_2005   | C:\MikeZero\...\Karup\Ka | Station 20.05 | C:\MikeZero\...\TIME\K | 20.05       | RMSE           | 1      | RMSE_Runoff   |
| 12 | RMSE_2006   | C:\MikeZero\...\Karup\Ka | Station 20.06 | C:\MikeZero\...\TIME\K | 20.06       | RMSE           | 1      | RMSE_Runoff   |

Figure 27.5 Objective Functions page

## 27.4.1 Output measures

### Name

Name of the output measure.

### Output file and item name

File name and corresponding item name of the time series of the simulation results at the observation point.

### Observation file and item name

File name and corresponding item name of the observation time series.

### Statistic type

AUTOCAL includes three basic comparison statistics:

Average error (*Avg. Error*):

$$AE = \frac{1}{N} \sum_{i=1}^N (OBS_i - SIM_i) \quad (27.1)$$



Root mean square error (*RMSE*):

$$RMSE = \sqrt{\frac{1}{N} \sum_{i=1}^N (OBS_i - SIM_i)^2} \quad (27.2)$$

Standard deviation of residuals (*St.Dev.*):

$$STD = \sqrt{\frac{1}{N} \sum_{i=1}^N (OBS_i - SIM_i - AE)^2} \quad (27.3)$$

where  $OBS_i$  and  $SIM_i$ ,  $i = 1, \dots, N$  are the observed and the corresponding simulated time series, respectively. Before calculation of the statistics, the time series are synchronised; that is, simulated values are extracted at the same time instants as the available observations using linear interpolation.

The three statistics are linked via the equation:

$$RMSE^2 = AE^2 + STD^2 \quad (27.4)$$

The statistic *AE* is a measure of the general offset between measurements and simulations (bias), whereas *STD* is a measure of the dynamical correspondence. *RMSE* is an aggregated measure that includes both bias and dynamical correspondence.

Besides the basic statistics, AUTOCAL includes two event-based statistics:

Error of maximum value (*Error of max.*):

$$ErrMax = Max\{OBS_i\} - Max\{SIM_i\} \quad (27.5)$$

Error of minimum value (*Error of min.*):

$$ErrMin = Min\{OBS_i\} - Min\{SIM_i\} \quad (27.6)$$

The maximum and minimum observed and simulated values are extracted in the period defined in the observation file.



### Weight

The weight assigned to the output measure in the objective function that aggregates several output measures. The assigned weights should reflect the measurement uncertainties, and the correlation between the measurements. That is, smaller weights should be given to more uncertain measurements, and if clusters of measurement points exist, these points should be given a lower weight than single point measurements in other parts of the modelling domain in order not to put undue emphasis on model performance in certain areas.

### Function name

The function name of the objective function in which the output measure should be included. In the drop-down menu the function names from the *Objective functions* table are given.

## 27.4.2 Objective functions

### Name

Name of the objective function.

### Function type

AUTOCAL uses three different functions for aggregation of the defined output measures:

Weighted sum:

$$F_{pool,i} = \sum_{j=1}^n w_j F_j \quad (27.7)$$

Weighted sum of absolute values:

$$F_{pool,i} = \sum_{j=1}^n w_j |F_j| \quad (27.8)$$

Weighted sum of squares:

$$F_{pool,i} = \sum_{j=1}^n w_j F_j^2 \quad (27.9)$$

where  $F_j$  is the output measure,  $w_j, j = 1, 2, \dots, n$  are the weights given to each measure, and  $n$  is the number of measures that are pooled.

Typically, output measures within a certain area that measure the same statistic for the same physical variable are pooled to evaluate the average model performance for that variable in the specified area with respect to bias (*Avg. Error*), dynamical behaviour (*St. Dev.*) or an overall goodness-of-fit (*RMSE*). The event-based statistics are typically pooled into an aggregate error of maximum and minimum values, respectively.

### Weight

The weight assigned to the objective function in the aggregation of the different objective functions into one aggregate measure to be optimised in the calibration. The assigned weights should reflect the relative priorities given to the different objectives, depending on the specific model application being considered. For investigating the entire Pareto front between the objective functions in a multi-objective calibration, the aggregated measure can be adopted by performing several optimisation runs using different weights.

## 27.4.3 Evaluation period

### Start date

The start date of the time series for which the output measures are calculated. It is generally recommended to set the start date after the start date of the model simulation in order to include a certain warm-up period in the simulation to minimise the influence from the initial conditions in the calculation of the output measures.

### End date

The end date of the time series for which the output measures are calculated. This is usually set to the end date of the model simulation.

## 27.4.4 Aggregation of objective functions

The defined objective functions are aggregated into one measure:

$$F = \sum_{i=1}^M w_i g_i(F_{pool,i}) \quad (27.10)$$

where  $M$  is the number of objective functions that are aggregated,  $w_i, i = 1, 2, \dots, M$  are the weights, and  $g_i(\cdot), i = 1, 2, \dots, M$  are transformation functions assigned to each objective function.



Three different transformation options are available:

No transformation:

$$g_i(F_{pool,i}) = F_{pool,i} \quad (27.11)$$

Transformation to a common distance scale:

$$g_i(F_{pool,i}) = \frac{F_{pool,i}}{\sigma_i} + \varepsilon_i \quad (27.12)$$

where  $\sigma_i$  is the standard deviation of the  $i$ 'th objective function of the initial population used in the Shuffled Complex Evolution or Population Simplex Evolution optimisation algorithm, and  $\varepsilon_i$  is a transformation constant given by:

$$\varepsilon_i = \max \left\{ \min \left\{ \frac{F_j}{\sigma_j}, j = 1, 2, \dots, M \right\} \right\} - \min \left\{ \frac{F_i}{\sigma_i} \right\} \quad (27.13)$$

Transformation to a common probability scale:

$$g_i(F_{pool,i}) = \Phi \left( \frac{F_{pool,i} - \mu_i}{\sigma_i} \right) \quad (27.14)$$

where  $\Phi(\cdot)$  is the cumulative distribution function of the standard normal distribution, and  $\mu_i$  and  $\sigma_i$  are the mean and the standard deviation of the  $i$ 'th objective function of the initial population.

The transformation functions that are applied in the transformation to a common distance scale and a common probability scale are introduced to compensate for differences in the magnitudes of the different measures so that all  $g_i(\cdot)$  have about the same influence on the aggregated objective function near the optimum. When using a population-based optimisation algorithm, such as the Shuffled Complex Evolution method and the Population Simplex Evolution, an initial population within the feasible region is evaluated. From this initial population, the transformation functions are calculated.



## 27.5 Scenario Runs

If the scenario run option has been chosen, the scenario run properties must be specified on the *Scenario Runs* page (Figure 27.6).

|   | Kh_Unit2 | Kh_Unit4 | DrainLevel | DrainConst | LeakCoef | Alpha_Soil1 | Ks_Soil1 | Alpha_Soil2 |
|---|----------|----------|------------|------------|----------|-------------|----------|-------------|
| 1 | 0.0001   | 1e-005   | -1         | 1e-007     | 1e-007   | 0.01        | 0.0001   | 0.05        |
| 2 | 0.0005   | 1e-005   | -1         | 1e-007     | 1e-008   | 0.05        | 1e-005   | 0.05        |
| 3 | 1e-005   | 5e-005   | -0.8       | 1e-008     | 1e-006   | 0.01        | 1e-006   | 0.1         |
| 4 | 0.0001   | 5e-005   | -0.8       | 1e-008     | 1e-007   | 0.05        | 1e-005   | 0.1         |
| 5 | 1e-005   | 1e-006   | -1.3       | 1e-006     | 1e-008   | 0.1         | 1e-006   | 0.01        |

Figure 27.6 Scenario Runs page

### 27.5.1 Scenario type

AUTOCAL includes two different options for performing scenario analysis:

- Single run using initial parameter values. In this case a single model run is performed using the initial parameter values given in the table on the *Model Parameters* page. When the model parameters and the objective functions have been specified, it is recommended to carry out a single run in order to check the setup
- Multiple runs. In this case multiple model runs are performed using the parameter values given in the *Parameters* table.

### 27.5.2 Parameters

In the *Parameters* table the set of variable parameters to be used in the scenario runs are specified.





## 27.6 Sensitivity Analysis

If the sensitivity analysis option has been chosen, the properties must be specified on the *Sensitivity Analysis* page (Figure 27.7).

Sensitivity analysis is often done as a first step in a model calibration to identify the most important model parameters to be fine-tuned in the succeeding parameter optimisation.

The screenshot shows a window titled "Sensitivity Analysis". Inside, there is a dropdown menu for "Sensitivity analysis method" set to "Local sensitivity analysis". Below this, there is a group box containing four settings: "Difference approximation" set to "Forward", "Perturbation option" set to "Fraction of parameter interval", "Perturbation fraction" set to "0.05", and a checked checkbox for "Calculate covariance matrix".

Figure 27.7 Sensitivity Analysis page

### 27.6.1 Sensitivity analysis method

The present version of AUTOCAL includes one sensitivity analysis method:

- Local sensitivity analysis

### 27.6.2 Local sensitivity analysis

Local sensitivity analysis provides the sensitivity of the model parameters around a specified parameter set, and hence gives information about the importance of the parameters only at that location in parameter space. If the simulation model is highly non-linear in its parameter-output interaction, sensitivity measures may vary considerably in the parameter space.



Thus, parameters that are insensitive for certain parameter sets may be highly sensitive for other parameter sets and vice versa.

The local sensitivity measures are calculated around the initial parameter set specified on the *Model Parameters* page.

### Difference approximation

The sensitivity of a parameter with respect to a model response (output measure) is defined as

$$S_i = \frac{\partial F}{\partial \theta_i} \quad (27.15)$$

where  $F$  is the output measure, and  $\theta_i$  is the considered model parameter. The sensitivity measure is evaluated around a specified parameter set  $(\theta_1, \theta_2, \dots, \theta_n)$ .

In AUTOCAL a finite difference approximation is used to evaluate the sensitivity coefficients. Three different options are available:

Forward difference approximation:

$$S_i = \frac{F(\theta_1, \theta_2, \dots, \theta_i + \Delta\theta_i, \dots, \theta_n) - F(\theta_1, \theta_2, \dots, \theta_n)}{\Delta\theta_i} \quad (27.16)$$

Backward difference approximation:

$$S_i = \frac{F(\theta_1, \theta_2, \dots, \theta_n) - F(\theta_1, \theta_2, \dots, \theta_i - \Delta\theta_i, \dots, \theta_n)}{\Delta\theta_i} \quad (27.17)$$

Central difference approximation:

$$S_i = \frac{F(\theta_1, \theta_2, \dots, \theta_i + \Delta\theta_i, \dots, \theta_n) - F(\theta_1, \theta_2, \dots, \theta_i - \Delta\theta_i, \dots, \theta_n)}{2\Delta\theta_i} \quad (27.18)$$

where  $\Delta\theta_i$  is the parameter perturbation.

The calculation of the sensitivity coefficients require  $n + 1$  model evaluations in the case of forward and backward difference approximations, and  $2n + 1$  model evaluations when the central difference approximation is applied.



### Perturbation option

The parameter perturbation can be calculated as:

A fraction of the initial parameter value:

$$\Delta\theta_i = f_c \theta_i \quad (27.19)$$

A fraction of the parameter interval:

$$\Delta\theta_i = f_c (\theta_{i, upper} - \theta_{i, lower}) \quad (27.20)$$

where  $\theta_{i, upper}$  and  $\theta_{i, lower}$  are the specified upper and lower limits of the parameter.

### Perturbation fraction

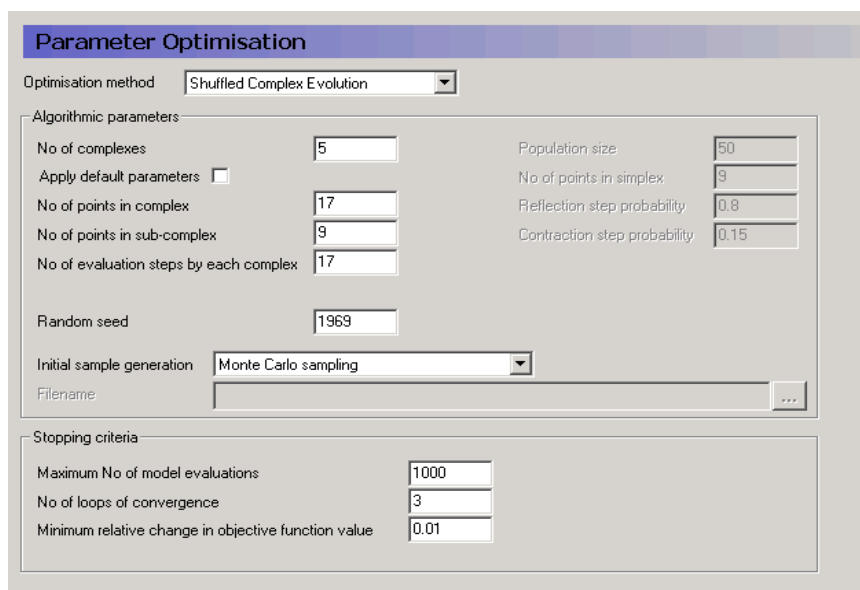
The perturbation fraction is the fraction  $f_c$  of the initial parameter value or the parameter interval depending on the choice of parameter perturbation.

### Calculate covariance matrix

If this option is selected, the covariance matrix of the parameters evaluated around the initial parameter set is calculated. This matrix is derived based on the sensitivities of the simulated values corresponding to each of the measurements with respect to each of the parameters. The matrix can only be calculated in the case a weighted least squares aggregated objective function is specified, i.e. each *Output measure* on the *Objective Functions* page is defined as a *RMSE* statistic, each *Objective function* is defined as a *Weighted sum of squares*, and the *Aggregation of objective functions* is set to *No transformation*.

## 27.7 Parameter Optimisation

If the parameter optimisation option has been chosen, the properties must be specified on the *Parameter Optimisation* page (Figure 27.8).



**Parameter Optimisation**

Optimisation method: Shuffled Complex Evolution

**Algorithmic parameters**

No of complexes: 5  
 Apply default parameters: ☐  
 No of points in complex: 17  
 No of points in sub-complex: 9  
 No of evaluation steps by each complex: 17  
 Population size: 50  
 No of points in simplex: 9  
 Reflection step probability: 0.8  
 Contraction step probability: 0.15  
 Random seed: 1969  
 Initial sample generation: Monte Carlo sampling  
 Filename:

**Stopping criteria**

Maximum No of model evaluations: 1000  
 No of loops of convergence: 3  
 Minimum relative change in objective function value: 0.01

Figure 27.8 Parameter Optimisation page with Shuffled Complex Evolution option chosen

### 27.7.1 Optimisation method

AUTOCAL includes two parameter optimisation methods:

- Shuffled Complex Evolution
- Population Simplex Evolution

### 27.7.2 Shuffled Complex Evolution method

The Shuffled Complex Evolution (SCE) method is a global optimisation algorithm that combines various search strategies, including (i) competitive evolution, (ii) controlled random search, (iii) the simplex method, and (iv) complex shuffling.

The SCE algorithm includes the following steps:

- 1 *Initialisation.* An initial sample of parameter sets  $\theta_i$  are randomly generated from the feasible parameter space defined by the lower and upper limits of each parameter on the *Model Parameters* page. For each parameter set the objective function value  $F_i = F(\theta_i)$  is calculated. The initial sample has the size  $s = pm$  where  $p$  is the number of complexes and  $m$  is the number of points in each complex.
- 2 *Partitioning into complexes.* The  $s$  points are ranked in order of increasing objective function value ( $F(1) < F(2) < \dots < F(s)$ ). The  $s$



points are partitioned into  $p$  complexes, such that points corresponding to function values  $\{F(1), F(p+1), \dots, F((s-1)p+1)\}$  form the 1<sup>st</sup> complex, points corresponding to function values  $\{F(2), F(p+2), \dots, F((s-1)p+2)\}$  form the 2<sup>nd</sup> complex, etc.

- 3 *Evolution.* A sub-complex of size  $q$  is formed from the complex by randomly choosing  $q$  points from the  $p$  points in the complex. A triangular probability distribution is used for assigning the probability of a point to be included in the sub-complex (i.e. larger probability for points with smaller objective function value). The sub-complex is evolved (offspring generation) according to the simplex algorithm. Each complex is evolved  $\beta$  times.
- 4 *Complex shuffling.* The new sample of  $s$  points is shuffled, cf. step 2.
- 5 Steps 2-4 are repeated until a stopping criterion is met.

### Algorithmic parameters

The algorithmic parameters of the SCE algorithm, their feasible range and recommended values are shown in Table 27.4.

*Table 27.4 Algorithmic parameters for the SCE algorithm ( $n$  = No. of calibration parameters), their range and recommended values*

| Parameter | Description   | Range             | Recommended value |
|-----------|---|-------------------|-------------------|
| $p$       | No. of complexes  | $p \geq 1$        | -                 |
| $m$       | No. of points in a complex                                    | $m \geq 2$        | $2n + 1$          |
| $q$       | No. of points in a sub-complex                                | $2 \leq q \leq m$ | $n + 1$           |
| $\beta$   | No. of evolution steps taken by each complex before shuffling | $\beta \geq 1$    | $2n + 1$          |

If one complex is chosen in SCE and the number of points in the complex as well as the sub-complex are set equal to  $n + 1$ , the local search simplex method is obtained as a special case.

### No. of complexes

Number of complexes  $p$  applied in the SCE algorithm. This is the most important parameter of the SCE algorithm. Sensitivity tests show that the dimensionality of the calibration problem (No. of calibration parameters)



is the primary factor determining the proper choice of  $p$ . In general, the larger value of  $p$  is chosen the higher the probability of converging into the global optimum but at the expense of a larger number of model simulations (the number of model simulations is virtually proportional to  $p$ ), and vice versa. One should choose  $p$  to balance the trade-off between the robustness of the algorithm and the computing time.

**No. of points in complex**

Number of points in a complex. A recommended value for this parameter is  $2n + 1$  where  $n$  is the number of calibration parameters.

**No. of points in sub-complex**

Number of points in a sub-complex. A recommended value for this parameter is  $n + 1$  where  $n$  is the number of calibration parameters.

**No. of evaluation steps by each complex**

Number of evaluation steps taken by each complex before complex shuffling. A recommended value for this parameter is  $2n + 1$  where  $n$  is the number of calibration parameters.

**Random seed**

Random seed used in the optimisation. Can be set to any positive integer value. Since the SCE method is a probabilistic search procedure, different optimisation results will be obtained by using different random seeds.

**Apply default parameters**

If the *Apply default parameters* option is selected, the default algorithmic parameters in Table 27.4 are used.

**Initial sample generation**

Three different options are available for generation of the initial sample in the SCE algorithm:

- Monte Carlo sampling. In this case the initial parameter sets are randomly generated within the feasible parameter range specified on the *Model Parameters* page assuming a uniform distribution.
- Latin hypercube sampling. In this case the individual parameters are sampled according to a stratified sampling scheme where the feasible parameter interval is divided into  $s$  equal intervals ( $s$  being the sample size) and a point is then randomly selected within each interval.
- Initial sample from previous optimisation run. This option allows continuing the optimisation from the last iteration loop of a previous optimisation run.

**File name**

File name of the file containing the optimisation results from a previous optimisation run to be used as initial conditions. The file to be specified is the AUTOCAL SCE optimisation output file (see Section 27.11.3).

**Stopping criteria**

Three stopping criteria are defined:

- Maximum number of model evaluations.
- Convergence in objective function space. In this case the optimisation terminates if the objective function of the best parameter set has not changed more than a user-defined minimum value in a given number of shuffling loops.
- Convergence in parameter space. In this case the optimisation terminates if the range of parameter values of the entire population in the parameter space is less than a given value (not user-defined).

The search terminates when one of these criteria is met.

**Maximum No. of model evaluations**

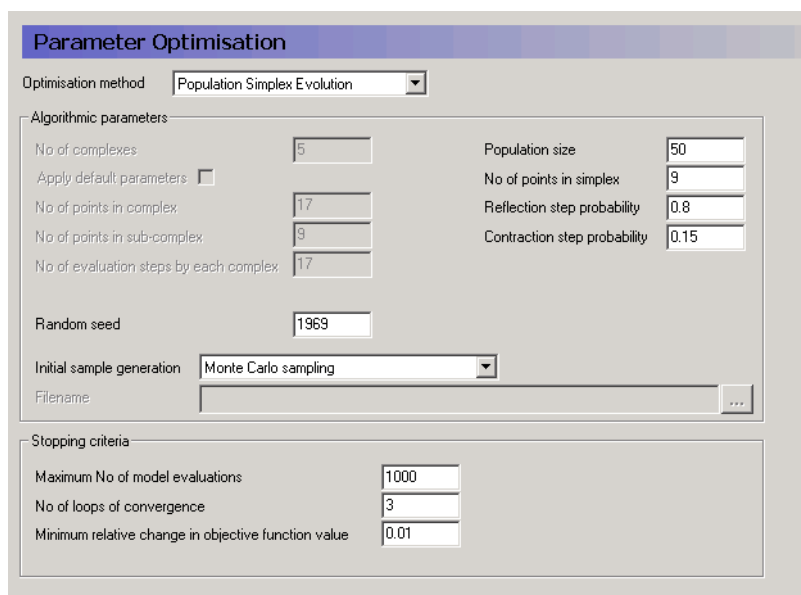
The maximum number of model evaluations allowed in the optimisation.

**No. of loops of convergence**

The number of iteration loops in which the objective function value of the best parameter set has not changed more than the *Minimum relative change in objective function value*.

**Minimum relative change in objective function value**

Minimum relative change allowed in the best objective function value in the last *No. of loops of convergence*.



**Parameter Optimisation**

Optimisation method: Population Simplex Evolution

**Algorithmic parameters**

|  |                          |                              |      |
|--|--------------------------|------------------------------|------|
| No of complexes                        | 5                        | Population size              | 50   |
| Apply default parameters               | <input type="checkbox"/> | No of points in simplex      | 9    |
| No of points in complex                | 17                       | Reflection step probability  | 0.8  |
| No of points in sub-complex            | 9                        | Contraction step probability | 0.15 |
| No of evaluation steps by each complex | 17                       |                              |      |

Random seed: 1969

Initial sample generation: Monte Carlo sampling

Filename:

**Stopping criteria**

|   |      |
|---|------|
| Maximum No of model evaluations                     | 1000 |
| No of loops of convergence                          | 3    |
| Minimum relative change in objective function value | 0.01 |

Figure 27.9 Parameter Optimisation page with Population Simplex Evolution option chosen

### 27.7.3 Population Simplex Evolution method

The Population Simplex Evolution (PSE) method is a global optimisation algorithm that is especially suited for parallel execution using the Office Grid facility in AUTOCAL. The method evolves a population of points using the reflection and contraction operators included in the simplex method. In addition a mutation component is added to minimise the risk of premature convergence.

The PSE algorithm can be summarised as follows:

- 1 *Initialisation.* An initial sample of size  $s$  of parameter sets  $\theta_i$  are randomly generated from the feasible parameter space defined by the lower and upper limits of each parameter on the *Model Parameters* page. For each parameter set the objective function value  $F_i = F(\theta_i)$  is calculated.
- 2 *Evolution.* From the population  $s$  simplexes of size  $q$  are formed. A triangular probability distribution is used for assigning the probability of a point to be included in the simplex (i.e. larger probability for points with smaller objective function value). The simplex is evolved according to the simplex operators and a mutation operator. Thus, a simplex can be evolved by reflection (with probability  $p_r$ ), contraction (with probability  $p_c$ ) or mutation (with probability  $1-p_c-p_r$ ). For each simplex a random value  $\varepsilon$  between 0 and 1 is generated and this value then





determines which operator should be applied for the evolution, i.e. reflection if  $\varepsilon < p_r$ , contraction if  $\varepsilon$  is in the interval  $[p_r, p_r + p_c]$ , and mutation if  $\varepsilon > p_r + p_c$ .

- 3 *Selection of new population.* The algorithm applies the principle of elitism to select the new population. The old population and the evolved population of points are sorted with respect to their objective function value. The best (i.e. smallest objective function value)  $s$  points defines the new population.
- 4 Steps 2-3 are repeated until a stopping criterion is met.

### Algorithmic parameters

The algorithmic parameters of the PSE algorithm, their feasible range and recommended values are shown in Table 27.5.

Table 27.5 Algorithmic parameters for the PSE algorithm ( $n$  = No. of calibration parameters), their range and recommended values

| Parameter | Description  | Range               | Recommended value |
|-----------|--|---------------------|-------------------|
| $s$       | Population size  | $s > n$             | -                 |
| $q$       | No. of points in a simplex                               | $2 \leq q < s$      | $n + 1$           |
| $p_r$     | Reflection step probability                              | $0 \leq p_r \leq 1$ | 0.70-0.90         |
| $p_c$     | Contraction step probability                             | $0 \leq p_c \leq 1$ | 0.05-0.20         |
| $p_m$     | Mutation probability determined as $p_m = 1 - p_r - p_c$ | $0 \leq p_m \leq 1$ | 0.01-0.10         |

### Population size

Population size  $s$  applied in the PSE algorithm. This parameter is important for the convergence properties of the algorithm. In general, the larger value of  $s$  is chosen the higher the probability of converging into the global optimum but at the expense of a larger number of required model evaluations. One should choose  $s$  to balance the robustness of the algorithm and the computing time. The proper choice of  $s$  depends on the dimensionality of the problem.

### No. of points in simplex

Number of points in a simplex. A recommended value for this parameter is  $n + 1$  where  $n$  is the number of calibration parameters.

**Reflection step probability**

The probability of performing a reflection step of the simplex. In this case the new point is found by reflecting the worst point of the simplex in the centroid of the remaining points. A recommended value for this parameter is in the range 0.70-0.90.

**Contraction step probability**

The probability of performing a contraction step of the simplex. In this case the new point is found as the mean between the worst point of the simplex and the centroid of the remaining points. A recommended value for this parameter is in the range 0.05-0.20. The sum of the reflection and contraction probabilities should be less than one. The remaining portion ( $1-p_r-p_c$ ) is assigned a mutation probability with a recommended value in the range 0.01-0.10.

**Random seed**

Random seed used in the optimisation. Can be set to any positive integer value. Different optimisation results will be obtained by using different random seeds.

**Initial sample generation**

The same three options are available for generation of the initial sample as in the SCE algorithm, i.e. Monte Carlo sampling, Latin hypercube sampling, and using a user-specified initial sample from a previous optimisation run.

**File name**

File name of the file containing the optimisation results from a previous optimisation run to be used as initial conditions. The file to be specified is the AUTOCAL PSE optimisation output file (see Section 27.11.3).

**Stopping criteria**

The same stopping criteria as used for the SCE algorithm are applied.

## **27.8 Save Output Files**

AUTOCAL allows output files from the individual model simulation runs to be saved. The names of the files to be saved are specified in the *Output files* table (Figure 27.10). The saved output files from the individual simulations are given an extension in the file name that contains the simulation number. For instance, if the simulation model produces a result file called



MyResults.dfs2, the saved files are named MyResults\_1.dfs2, MyResults\_2.dfs2, etc.

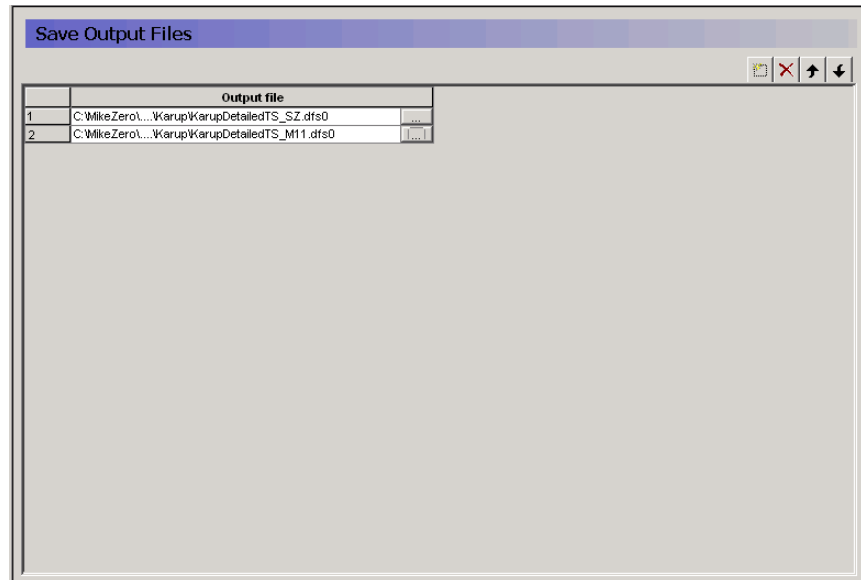


Figure 27.10 Save Output Files page

## 27.9 Office Grid

AUTOCAL includes an Office Grid extension for distributed computing whereby the calculations can be run on a number of PCs connected in a network. This includes also execution on multi-processor PCs. To use this facility the OfficeGRID software must be installed on all computers to be used in the network. The OfficeGRID is configured by defining a primary server that acts as the master for the AUTOCAL Office Grid application and a number of clients that do the model evaluations. Note that an OfficeGRID primary server can also be used as a client in the OfficeGRID network.

AUTOCAL Office Grid can be used with all the simulation options. In the case of *Scenario runs* or *Sensitivity analysis* all the parameter sets to be evaluated by the model are submitted to the OfficeGRID network for execution at the beginning of the simulations. When all parameter sets are evaluated on the OfficeGRID clients, the AUTOCAL simulation is finished. In the case of *Parameter optimisation* only the *Population Simplex Evolution* algorithm can be applied for parallel execution. For each iteration step in the algorithm the population of parameter sets to be evaluated are submitted to the OfficeGRID network for execution. When all the



parameter sets of the population are evaluated on the OfficeGRID clients, the master processes the results and determines a new population of parameter sets to be evaluated, which are then submitted to the OfficeGRID network for execution. The AUTOCAL simulation is finished when one of the stopping criteria of the algorithm has been reached.

The AUTOCAL result files from the simulation are saved on the PC acting as the AUTOCAL master. However, the model output files specified on the *Save Output Files* page that are saved from the individual model simulation runs are saved on the client PCs.

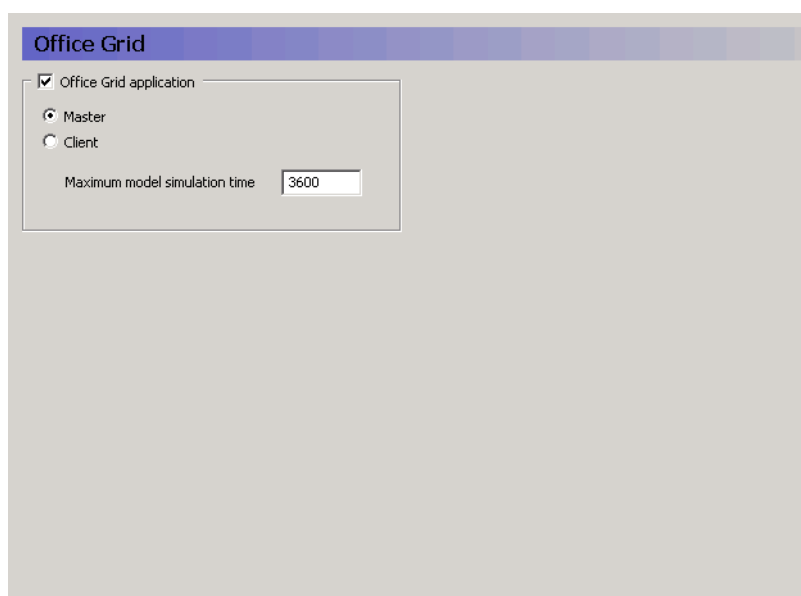


Figure 27.11 Office Grid page

### Office Grid application

If the *Office Grid application* option is selected, the AUTOCAL simulation is to be run on a network of PCs using the OfficeGRID facilities.

#### Master

For the AUTOCAL Office Grid application one *Master* is defined. This master must be executed on the PC that has been configured as an OfficeGRID primary server. The AUTOCAL master handles the AUTOCAL simulation and submit jobs (parameter sets to be evaluated) to the OfficeGRID network.



### Client

For the AUTOCAL Office Grid application a number of *Clients* can be defined. A client must be executed on a PC that has been configured as an OfficeGRID client or an OfficeGRID primary server. The AUTOCAL client executes the jobs (parameter sets) submitted by the AUTOCAL master.

### Maximum model simulation time

The *Maximum model simulation time* is the maximum time allowed for one model simulation on the OfficeGRID network. When a client executes a model simulation, the AUTOCAL master checks the used simulation time, and in the case the time exceeds the *Maximum model simulation time* the job is resubmitted to the OfficeGRID network for execution on another client. This ensures that the AUTOCAL simulation can continue if a client for some reason becomes disconnected from the network or if it is busy with other processes.

## 27.10 Start AUTOCAL Simulation

To start an AUTOCAL simulation choose:

Run | Simulation

See also Figure 27.12.

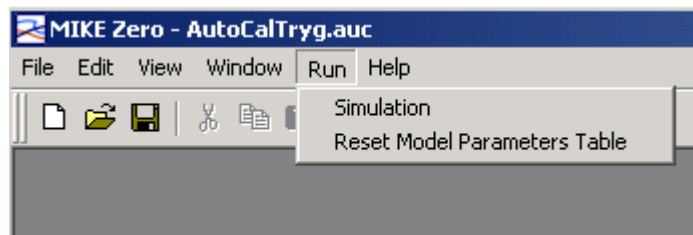


Figure 27.12 Start AUTOCAL simulation

## 27.11 Output

### 27.11.1 General output files

AUTOCAL produces two general output files:



- A log file that contains summary information about the AUTOCAL simulation. The log file is named MyAutocalRun.log where MyAutocalRun is the user-specified name for the AUTOCAL .auc input file. The file is saved in the working directory.
- A file that contains results from the individual model simulations, including the evaluated parameter set and the calculated output measures and objective functions. The file is named MyAutocalRun.dat and is saved in the working directory. The result file is tab delimited and can be directly imported into a spreadsheet.

### 27.11.2 Sensitivity analysis output

If the sensitivity analysis option is chosen, AUTOCAL produces a result file that contains the calculated sensitivity coefficients of each parameter with respect to the different output measures and objective functions. The file is named MyAutocalRun\_Sensitivity.dat and is saved in the working directory. The result file is tab delimited and can be directly imported into a spreadsheet.

#### Local sensitivity analysis

In the case of local sensitivity analysis the result file contains the following:

- The parameter set and corresponding output measures and objective functions for the point where the local sensitivity analysis has been performed.
- Scaled sensitivity coefficients for each parameter of the specified output measures and objective functions.
- Parameter covariance matrix in terms of the standard deviations of the parameters and the correlation matrix. It should be noted that the calculated parameter covariance is related to the transformed parameter values and not their native values.

In order to compare the local sensitivity coefficients between parameters of different scales of magnitude scaled sensitivity values are calculated as

$$S_{i, scale} = S_i(\theta_{i, upper} - \theta_{i, lower}) \quad (27.21)$$

where  $S_i$  is the calculated un-scaled sensitivity coefficient described in Section 27.6.2, and  $\theta_{i, upper}$  and  $\theta_{i, lower}$  are the specified upper and lower limits of the parameter. The scaled sensitivities provide a ranking of the parameters with respect to the importance of the parameters for the con-



sidered output measure or objective function. Higher scaled sensitivity values (absolute values) indicate more sensitive parameters.

As a rule of thumb, parameters are said to be insensitive if their scaled sensitivity value is less than about 0.01-0.02 times the maximum scaled sensitivity value (absolute value). However, care should be taken using this as a strict threshold measure. Since the sensitivity coefficients are evaluated only around the initial parameter set, they reflect the local sensitivities only. At other locations in the parameter space the sensitivity coefficients may be very different, especially if the simulation model is highly non-linear in its parameter-output interaction.

### **27.11.3 Optimisation output**

If the parameter optimisation option is chosen, AUTOCAL produces a result file that contains the results of the optimisation. The file is named `MyAutocalRun_Optimisation.dat` and is saved in the working directory. The result file is tab delimited and can be directly imported into a spreadsheet.

In the log file a summary of the optimisation results is written, including the best parameter set and corresponding objective function value for each iteration loop.

When the stopping criterion of the optimisation run is met, AUTOCAL performs a final model run using the optimised parameters. The result of this run is written in the last line of the `MyAutocalRun.dat` file. The model parameter files contain the optimised parameter values.

### **Shuffled Complex Evolution and Population Simplex Evolution**

For each iteration loop performed in the SCE or PSE optimisation the result file `MyAutocalRun_Optimisation.dat` contains the population of parameter sets and corresponding objective functions sorted with respect to the aggregated objective function value. This file should be used as the initial sample file for a subsequent SCE or PSE optimisation that continues the optimisation.

## **27.12 Error handling**

When AUTOCAL performs a model simulation, the return code from the called program is checked. In the case an error in the model simulation has occurred for a given parameter set, e.g. by creating instabilities, an error return code is received by AUTOCAL (a return value different from 0). In such a case the output measures and objective functions are given a delete value of -99, which are written to the output files.



Depending on the simulation option chosen in AUTOCAL, error return codes from the called program are handled differently:

- In the case the *Scenario runs* option is chosen AUTOCAL continues with the next scenario.
- In the case the *Sensitivity analysis* options is chosen AUTOCAL is halted. In this case the sensitivity coefficients and the parameter covariance matrix cannot be calculated.
- In the case the *Parameter optimisation* option is chosen, the optimisation continues and the parameter set that gave an error in the simulation model is penalised in the optimisation by assigning the aggregated objective function a penalty value of  $10^{30}$ . However, if a large number of model runs return error codes, a general model failure is suspected and AUTOCAL is halted.

If the simulation model does not provide a return code, AUTOCAL offers another possibility for error handling. When a model run is started, AUTOCAL creates an empty file called *AutocalSimErr.dat* that is placed in the working directory. When the model run finishes, AUTOCAL checks the file, and if it is not empty it is assumed that the model simulation has failed. In this case AUTOCAL handles the simulation in the same way as when it receives an error return code as described above.

To use this facility, it is necessary to include as part of the model simulation an option that writes an error message (or any dummy string) to the *AutocalSimErr.dat* file if an error occurs during the model simulation. If an error message is written to the file, the file can be saved as part of the AUTOCAL run for later inspection by using the *Save Output Files* option.





*Phi Software*

## **INDEX**



- A**  
Actual time step  
    ET . . . . . 72  
    OL . . . . . 72  
    UZ . . . . . 72  
Aggregate measure . . . . . 374  
ASCII File . . . . . 270, 311  
Auto Updater . . . . . 21  
Axis information . . . . . 257
- B**  
Batch Files . . . . . 68  
    Create . . . . . 68  
    Results . . . . . 69  
    Setup . . . . . 68  
Bilinear Interpolation . . . . . 292
- C**  
Calculator . . . . . 275, 336  
Comparison statistics . . . . . 371  
Complexes . . . . . 380  
Constant parameter . . . . . 368  
Coupling MIKE 11 to MIKE SHE . . 163  
    All branches . . . . . 169  
    Bed Leakage . . . . . 174  
    Bed Topography . . . . . 173  
    Exchange Type . . . . . 170  
    Flood Area . . . . . 172  
    Leakage Coefficient . . . . . 170  
Coupling MIKE 11 to MIKE SHEFlood  
    Code . . . . . 173  
Coupling MIKE SHE to MOUSE . . 183  
    Output Files . . . . . 186  
    Run . . . . . 187  
Crop . . . . . 341
- D**  
Define  
    Grid . . . . . 39  
    Model domain . . . . . 39  
Dependent parameter . . . . . 368  
dfs2 file . . . . . 40
- E**  
Editing modes . . . . . 261
- Equation parser . . . . . 369  
EUM Data Units . . . . . 349  
    Change . . . . . 352  
    Default units . . . . . 352  
    Imperial . . . . . 351  
    SI . . . . . 351  
Evolution . . . . . 381  
Export to ASCII File . . . . . 273  
Extra Parameters . . . . . 143
- F**  
File Formats . . . . . 269, 307  
File Properties Dialog . . . . . 256  
Filter . . . . . 335  
Finite difference approximation . . 378  
Flow Model . . . . . 27  
Flow Storing Requirements . . . . 228  
Font Settings Dialog . . . . . 268
- G**  
General Information . . . . . 257  
Geologic Model . . . . . 47  
    Conductivity values . . . . . 49  
    Lenses . . . . . 50  
GeoViewer . . . . . 158  
Graphical and font settings . . . . 262  
Graphical Settings . . . . . 267  
Graphical View . . . . . 260, 322  
Grid Editor  
    2D to 3D Layer Mapping . . . . 299  
    File to Copy . . . . . 298  
    Item mapping . . . . . 298  
    Operation . . . . . 299  
    Sub-area position . . . . . 299  
    Target file . . . . . 298  
    Time Position . . . . . 299
- H**  
Horizontal Interpolation . . . . . 49
- I**  
Identification tags . . . . . 366  
Integer Grid Codes . . . . . 290  
    Grid File (.dfs2) . . . . . 290  
    Polygons . . . . . 291



|                                       |          |                                      |          |
|---------------------------------------|----------|--------------------------------------|----------|
| Uniform Value . . . . .               | 290      | <b>R</b>                             |          |
| Interpolation . . . . .               | 276      | Redistribution of ponded water . . . | 156      |
| Introduction . . . . .                | 303      | Results                              |          |
| Item information . . . . .            | 258      | Time Series Output . . . . .         | 59       |
| <b>L</b>                              |          | <b>S</b>                             |          |
| Latin hypercube sampling . . . . .    | 382      | Scaled sensitivity values . . . . .  | 390      |
| Log file . . . . .                    | 390      | Select Sub-Set . . . . .             | 277      |
| Logarithmic transformation . . . . .  | 369      | Sensitivity measures . . . . .       | 378      |
| <b>M</b>                              |          | Setup                                |          |
| Measurement uncertainties . . . . .   | 373      | Data Tree . . . . .                  | 35       |
| MIKE 11 result file . . . . .         | 363      | Sheet Pile Module . . . . .          | 143      |
| MIKE GeoModel . . . . .               | 158      | Flux Units . . . . .                 | 195      |
| MIKE Zero Toolbox . . . . .           | 363      | Input File . . . . .                 | 145      |
| Monte Carlo sampling . . . . .        | 382      | Leakage Coefficient . . . . .        | 144      |
| MOUSE . . . . .                       | 179      | Location . . . . .                   | 144      |
| MsheMouse.pfs file . . . . .          | 184      | shp file . . . . .                   | 40       |
| Multi-objective calibration . . . . . | 374      | Simulation Parameters . . . . .      | 214      |
| <b>N</b>                              |          | Solute Transport . . . . .           | 195      |
| Navigation . . . . .                  | 331      | Calibration . . . . .                | 193      |
| New File . . . . .                    | 251, 309 | Concentration . . . . .              | 195      |
| Number of complexes . . . . .         | 381      | Requirements . . . . .               | 192      |
| <b>O</b>                              |          | Results . . . . .                    | 192      |
| Overland Flow                         |          | Verification . . . . .               | 193      |
| Boundary Conditions . . . . .         | 45       | Spatial Data                         |          |
| Overland Flow Routing                 |          | Stationary Real Parameters . . .     | 283      |
| Directly to the river . . . . .       | 156      | Time Varying Real Parameters .       | 283      |
| <b>P</b>                              |          | Species DEPENDENT Data . . . .       | 206      |
| Palette . . . . .                     | 322      | Species INDEPENDENT Data . . .       | 199      |
| Parameter perturbation . . . . .      | 379      | Stationary Real Parameters . . . .   | 283      |
| Pareto front . . . . .                | 374      | Statistics . . . . .                 | 278      |
| Partickle Tracking                    |          | Stopping criteria . . . . .          | 383      |
| Execute . . . . .                     | 232      | Storing Time Steps . . . . .         | 72       |
| Particle Tracking . . . . .           | 227      | <b>T</b>                             |          |
| Post-processing of model results . .  | 363      | Tabular View . . . . .               | 260, 321 |
| PT Output Retrieval . . . . .         | 234      | Template file . . . . .              | 365      |
| Limitations . . . . .                 | 235      | Time Varying Real Parameters . .     | 283      |
| PT Simulations . . . . .              | 228      | Toolbars . . . . .                   | 328      |
| Output . . . . .                      | 232      | Transformation functions . . . . .   | 374      |
| PT_OR File . . . . .                  | 235      | Triangular Interpolation . . . . .   | 296      |
| Example . . . . .                     | 236      | TS Types graphical representation    | 262      |
|                                       |          | TSF file . . . . .                   | 196      |
|                                       |          | Example . . . . .                    | 219      |

**U**

## Unsaturated and Saturated Zone

|                             |    |
|-----------------------------|----|
| Coupling . . . . .          | 46 |
| UZ Classification . . . . . | 45 |

**V**

|                                  |     |
|----------------------------------|-----|
| Variable parameter . . . . .     | 368 |
| Veritcal Interpolation . . . . . | 49  |

**W**

## Water Balances

|   |     |
|---|-----|
| Batch Mode . . . . .                      | 129 |
| Custom . . . . .                          | 139 |
| Standard Types . . . . .                  | 136 |
| Types . . . . .                           | 130 |
| Weighted sum . . . . .                    | 373 |
| Weighted sum of absolute values . . . . . | 373 |
| Weighted sum of squares . . . . .         | 373 |
| Well Fields . . . . .                     | 228 |
| WM Requirements . . . . .                 | 227 |

**X**

|                     |     |
|---------------------|-----|
| XTSF file . . . . . | 212 |
|---------------------|-----|

**Z**

|                |     |
|----------------|-----|
| Zoom . . . . . | 261 |
|----------------|-----|